មាតិកា

១. មេរៀនទី១: ការគណនាធ្នឹមបេតុងអាមេ (RC Beam Design)……1

1. ការកំណត់ Beam Definition និង Beam Section ……6
2. ការកកំណត់ប្រហោងរបស់ធ្នឹមដើម្បីដាក់ខ្សែភ្លើង ឬ ទុយោ (Beam opening)……9
3. ការកំណត់ការដាក់បន្ទុកលើធ្នឹម (Beam Loads)……..10
4. ការកំណត់ Story Parameters ……13
5. ការកំណត់ Calculation Option ……13
6. ការកំណត់ Reinforcement Patterns ……15
7. ចាប់ផ្តើមគណនារកសៃដែក Start Calculation……..18
8. ការត្រួតពិនិត្យមើលការដាក់រៀបចំសៃដែក Beam- Reinforcement ……22
9. កំណត់សំគាល់ការគណនា Calculation Note ……23
10. Drawing Parameter ……..23
11. មើលប្លង់ Drawings ……25

២. មេរៀនទី២: ការគណនាកម្រាលខណ្ឌ (RC slab design) ……26

1. ការកំណត់ Code Units and Formats ……...29
2. កំណត់ Materials ត្រូវយកកូដ Euro – Code ……..30
3. កំណត់ Steel ……...30
4. កំណត់ Concrete ……30
5. កំណត់ Database for reinforcement ……31
6. កំណត់ Design Code ……..32
7. កំណត់ Load ក្នុងប្រអប់ Load Combination ……32

I. ការកំណត់ការគូសកម្រាលខណ្ឌ ……33

II. ការកំណត់កំរាស់កម្រាលខណ្ឌ Slab thickness ……...34

III. ការកំណត់ទំរ និង ការដាក់បន្ទុក Determine supported ……....38

IV. ការបង្កើតប្រភេទបន្ទុក Load Types ……..39

VI. កំណត់ការដាក់បន្ទុកលើកម្រាលខណ្ឌ Apply Loads ……...39

VII. ការបន្សំបន្ទុក Combination Loads ……..……42

VIII. ការ Mesh កម្រាលខណ្ឌ ……45

IX. ការគណនាកម្រាលខណ្ឌ ……47

X. ការគណនាសសៃដែក ……50

មេរៀនទី៣: ការគណនាសសរ RC Column Design ……64

I. ការកំណត់ Tools ……..67

1. ការកំណត់ Units and Formats ……67
2. ការកំណត់ Materials ……..68
3. ការកំណត់ Steel….. ……...68
4. ការកំនត់ Concrete ……68
5. ការជ្រើសរើស Database ……69
6. កំណត់ Design code ……..70
7. ការកំណត់ Load ក្នុងប្រអប់ Code combination ……70

មេរៀនទី៤: ការគណនាគ្រឹះរាក់ Shallow Foundation Design ……84

I. កំណត់ Job Preperence …….....87

1. ការកំណត់ Units and Formats ……..87

2. កំណត់ Code Materials ……88

3. កំណត់ Tab steel ……88

4. កំណត់ Tab Concrete ……..88

5. កំណត់ Database ……89

6. កំណត់ Load ក្នុងប្រអប់ Code Combinations ……90

II. ការកំណត់ទំហំគ្រឹះ សសរ និង ប្រភេទគ្រឹះ Foundation Definition……..91

1. ការកំណត់ទំហំជើងតាង និង សសរត្បឺ ……91

2. ការកំណត់ទំហំសសរ Tab Pier …….. 91

III. ការកំណត់ស្រទាប់ដីនៅក្រោមគ្រឹះ Foundation Soil ……..92

1. ជ្រើសរើស Icon Foundation soil ……92

2. ជ្រើសរើសប្រភេទដីបន្ថែម Edit database ……..92

IV. ការកំណត់បន្ទុកដែលមានអំពើមកគ្រឹះ Foundation Loads ……93

1. ជ្រើសរើស Icon Foundation Load ……93

2. ការកំណត់បន្ទុកថេរ Dead Load ……..94

3. ការកំណត់បន្ទុកអថេរ Live Load ……94

V. ការកំណត់ Story Parameter ……95

VI. ការកំណត់ Geotechnical Option ……95

VII. ការកំណត់ Calculation Option ……96

VIII. ការកំណត់ Reinforcement Pattern …….. 98

IX. ការចាប់ផ្តើមគណនារកសរសៃដែក Start Calculation ……99

X. ការត្រួតពិនិត្យមើលការរៀបចំសរសៃដែក Foundation Reinforcement ……..100

www.yerngshare.blogspot.com

XI. Calculation Note ……101
XII. Drawing Parameters ……101
XIII. Drawing ……..102

មេរៀនទី៥: ការគណនាក្នុងបួង 3D ……103

1. ការកំណត់យកអ័កស Axis សំរាប់ការគូរ ……103

1.1 ការកំណត់អ័ក្ស X ……..103

1.2 ការកំនត់អ័ក្ស Y……104

1.3 ការកំនត់អ័ក្ស Z ……105

2. ការគូសធ្នឹម និង សសរ ……..107

2.1 ការបង្កើតមុខកាត់ធ្នឹម ……107

2.2 ការបង្កើតមុខកាត់សសរ ……..108

2.3 ការកំណត់បន្ទុកដែលមានអំពើលើធ្នឹម ……109

មេរៀនទី៦: Truss 2D Design ……122

1. ការកំណត់ Material និង Code សម្រាប់គណនា ……..123

2. ការកំណត់ទំរង់របស់ Truss ……..125

3. ការកំណត់ទំរ Truss ……127

4. ការកំណត់បន្ទុកដែលមានអំពើលើ Truss ……128

4.1 ការកំណត់ប្រភេទបន្ទុក Load Types ……..128

4.2 ការកំណត់តំលៃបន្ទុក ……128

4.3 ការដាក់បន្ទុក DL1 ……..128

4.4 ការដាក់បន្ទុក LL1 ……..129

4.5 ការបន្សំបន្ទុក Load Combination ……..130

www.yerngshare.blogspot.com

5. ការកំណត់រកកំលាំងក្នុងរបាររបស់ Truss ……..131

6. ការត្រួតពិនិត្យ Check ឬ Design មុខកាត់របាររបស់ Truss ……...133

6.1 Member verification ……133

មេរៀនទី៧: 3D Steel Frame ……...134

1. Click Icon Frame 3D Design ……...136

2. ការកំណត់អ័ក្ស Axis សំរាប់គូរ ……...136

2.1 កំណត់អ័ក្ស X ……137

2.2 កំណត់អ័ក្ស Y ……137

2.3 កំណត់អ័កស Z ……137

3. ការគូសសរ និង Steel Frame ……..139

4. ការគូសសរបេតុងអាមេ ……..140

5. ការគូសសរ UPA 160 ……...140

6. ការដាក់ទំរ Support ……141

7. ការគូសធ្នឹម Beam ……141

8. ការថតចំលងរូបភាព Copy Frame ……142

9. ការគូស CAE 40 x 3 ……143

10. គូសរបារដែកដែលមានមុខកាត់ CAE 40 x 3 ……143

11. ការកំណត់បន្ទុកមានអំពើលើធ្នឹម 155

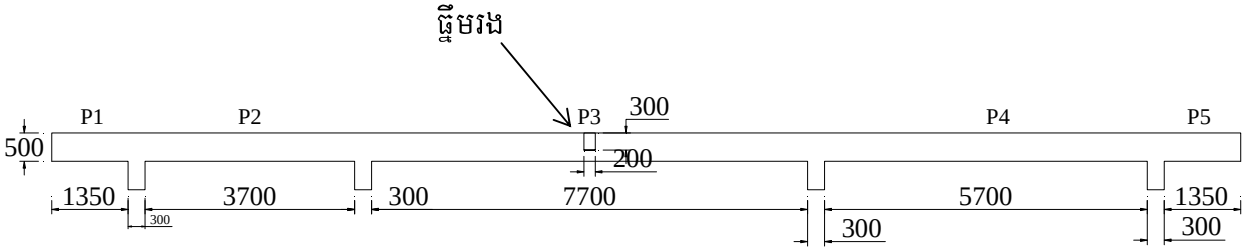
12. Reference ……..156

www.yerngshare.blogspot.com

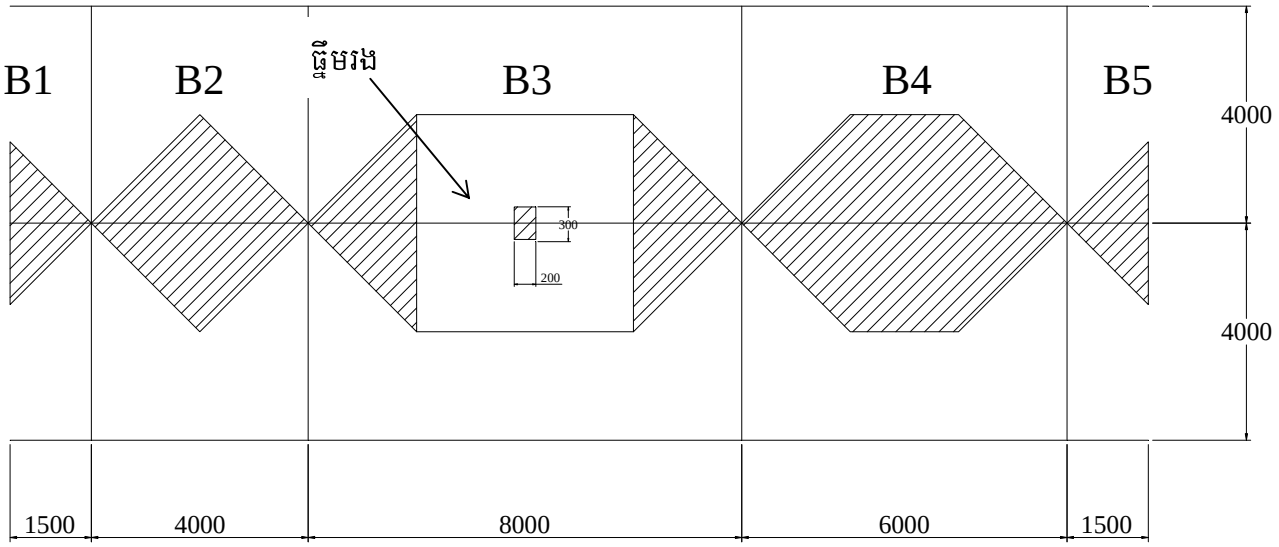
មេរៀនទី១

ការគណនាធ្នឹមបេតុងអាម៉េ

RC Beam Design



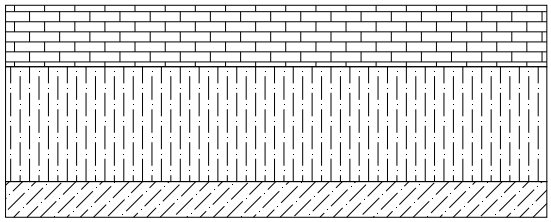
<u>សំគាល់</u> ៖ យើងត្រូវបែងចែកមុំ ៤៥ ដឺក្រេ ទៅលើធ្នឹមដែលយើងត្រូវដាក់បន្ទុកពីលើ ។

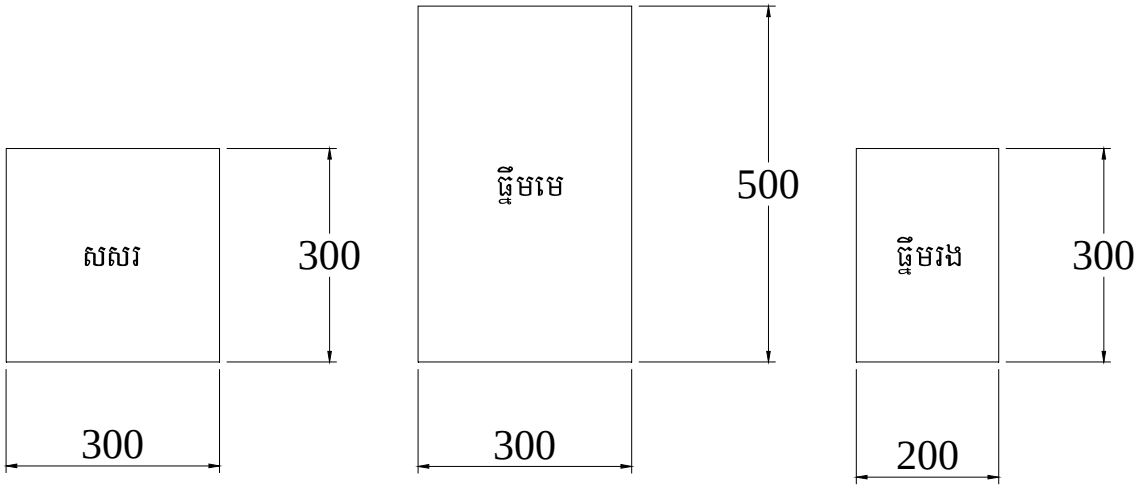


Plan View

មុខកាត់គ្រឿងបង្គុំ Frame Section :

- មុខកាត់ធ្នឹមមេ : 30cm x 50cm
- មុខកាត់ធ្នឹមរង : 20cm x 30cm
- មុខកាត់សសរ : 30cm x 30cm
- ការ៉ូ និង បាយអរ : 5cm (Mortar + Tile)
- បង្គប់សេកំរាស់ : 10cm (Slab)
- បាយអរបូកក្រោមបង្គប់សេ : 1.5cm (Plastering)





បន្ទុកមានអំពើមកលើកំរាលខណ្ឌក្នុង 1 m^2 :

- Self - Weight (មិនចាំបាច់កំណត់ទេ ពីព្រោះទុកអោយ Program វាកំណត់ដោយខ្លួនវា)
- Live Load : LL=1.5 kN/m^2
- Dead Load : (Slab + Mortar + Tile + Plastering)

  Slab = 0.1m x 25kN/m^3 = 2.5kN/m^2

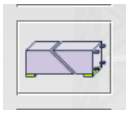
  Mortar + Tile + Plastering = (0.05m + 0.015m) x 20kN/m^3 = 1.2kN/m^2
- Total Dead Load : 3.7kN/m^2

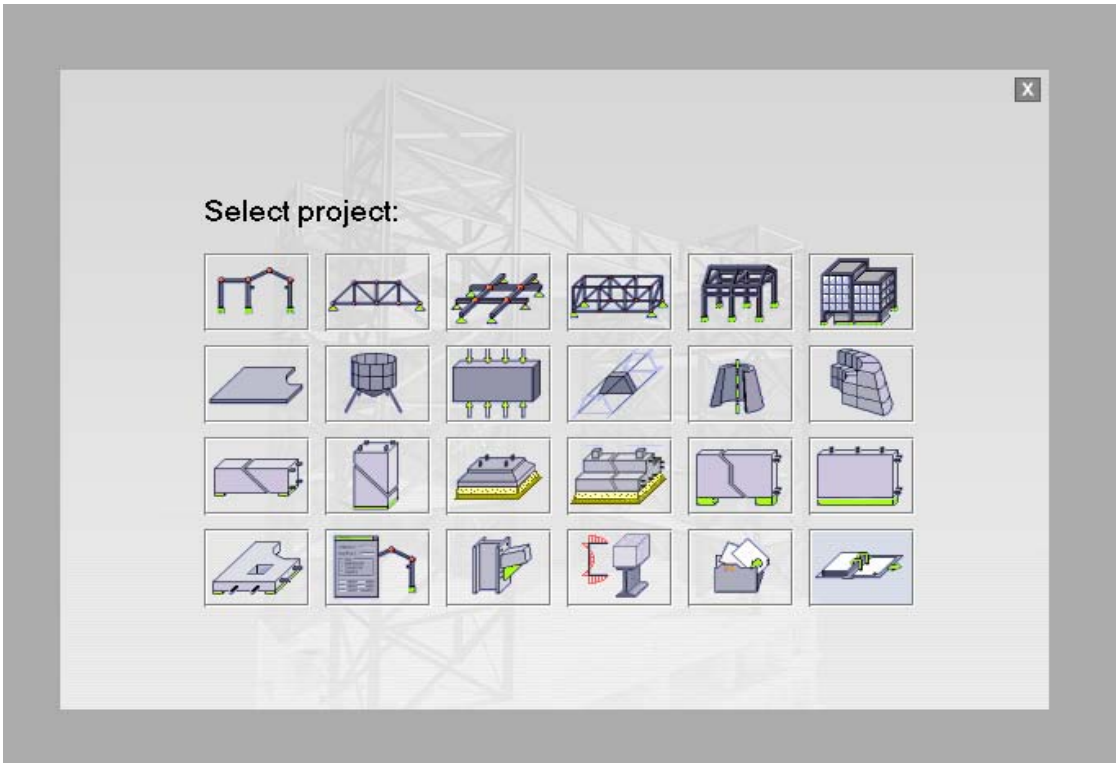
Analysis and Design ដោយ : ACI code

Material Properties for Analysis and Design :

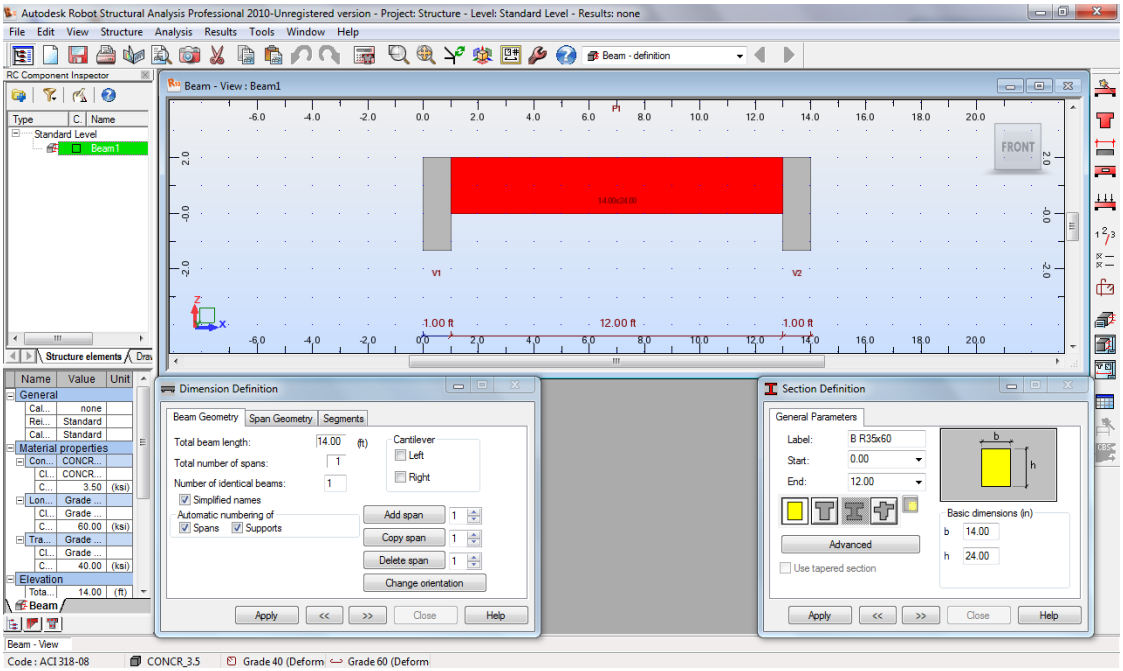
- Yield Strength of Longitudinal Reinforcement : 390 MPa
- Yield Strength of Transversal Reinforcement : 235 MPa
- Compressive Strength of Concrete (Cylinder) : 25 MPa

Step by Step Solution

នៅពេលបើក Program Autodesk Robot Structural Analysis Professional 2010 ដំបូងយើងនិងឃើញដូចរូបខាងក្រោមបន្ទាប់មកយើងជ្រើសរើសយក Icon  ដូចដែលបានបង្ហាញនៅលើរូបខាងក្រោម ៖

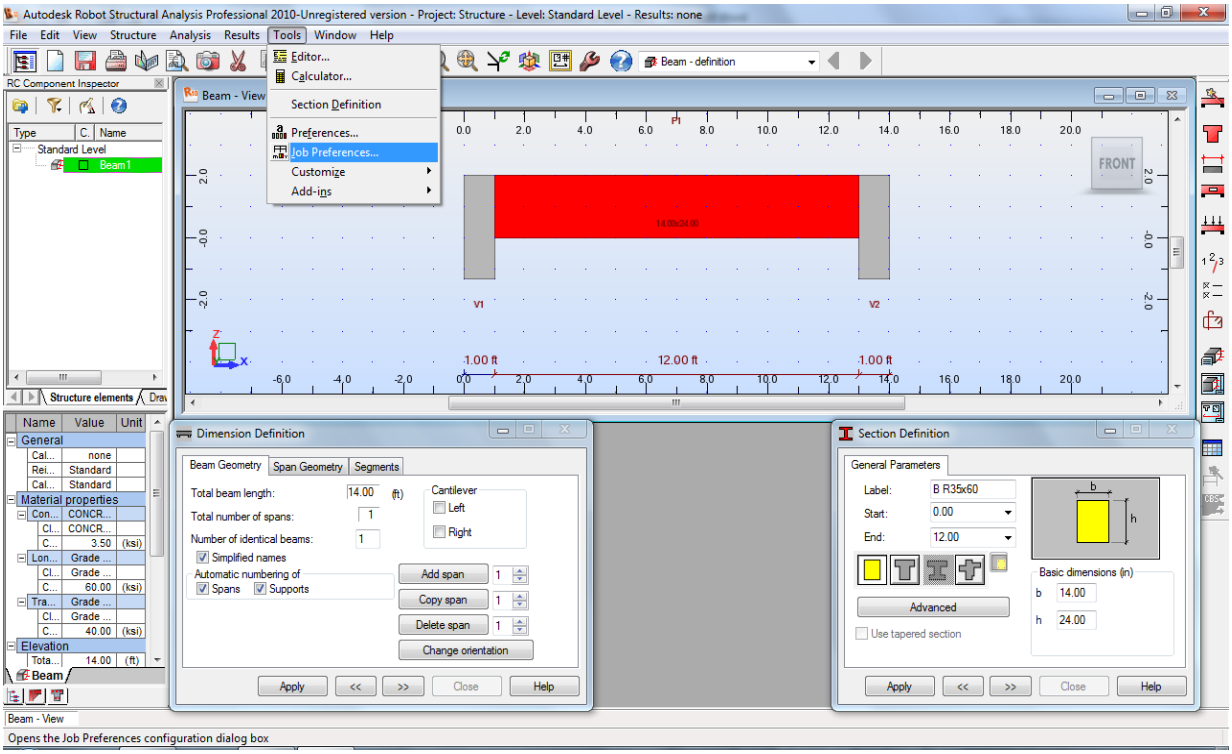


- បន្ទាប់មកទៀតអេក្រង់នឹងបង្ហាញដូចរូបខាងក្រោមនេះ

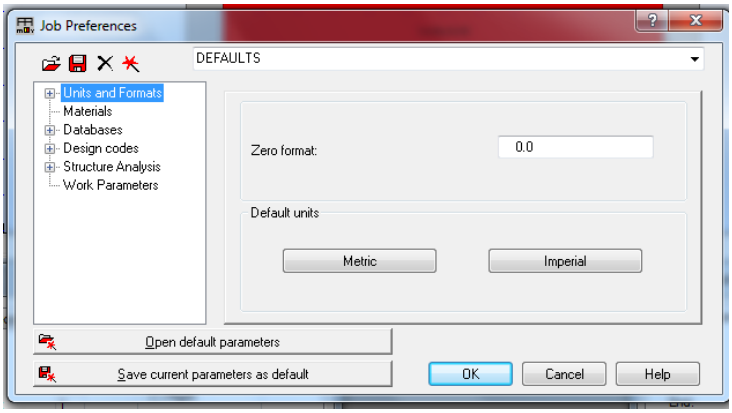


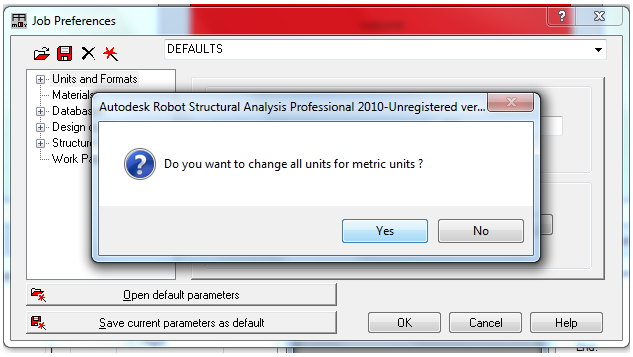
- **<u>បន្ទាប់មកចូរកំណត់</u>** <u>Units and Format, Materials and Codes មុនពេលធ្វើការ Analysis and Design :</u>

  - ចុចលើ Tool Menu > Job Preference……

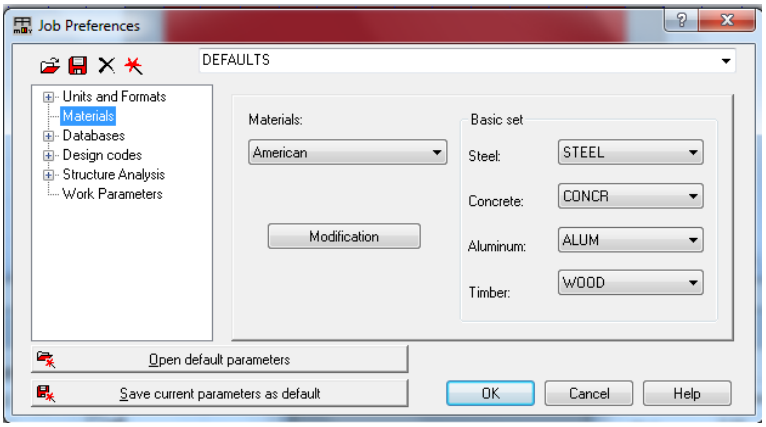


  - អេក្រង់នឹងបង្ហាញផ្ទាំងដូចរូប នៅត្រង់ Units and Formats ត្រូវកំណត់វាជា Metric រួច Yes

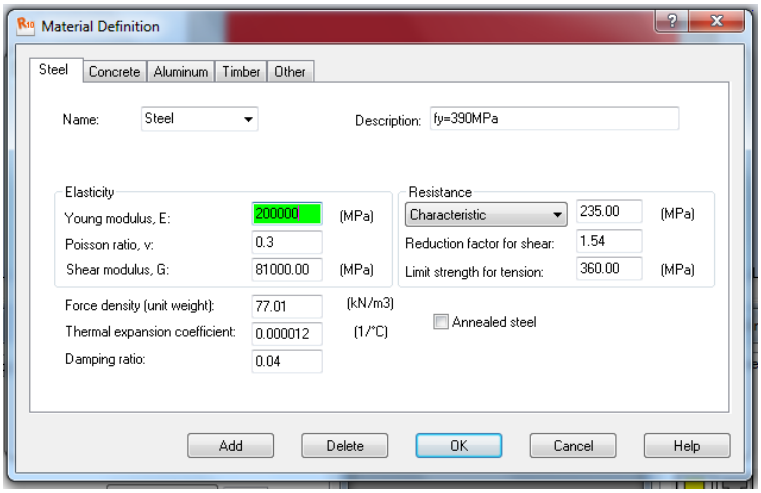




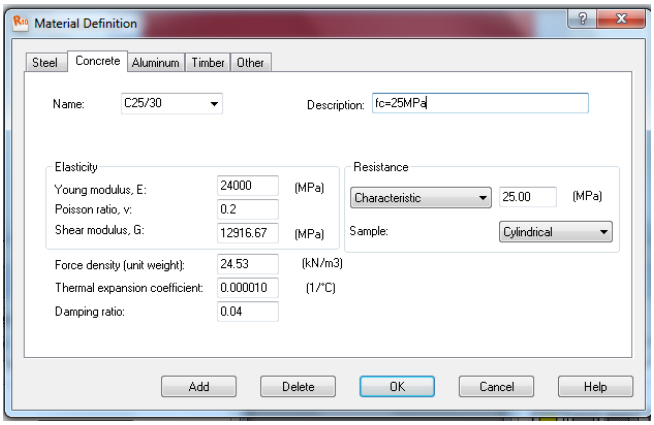
- នៅត្រង់ Materials ត្រូវកំណត់កូដអោយ Materials ត្រូវយកកូដ Euro code រួចចុចទៅលើ Modification ដើម្បីកំណត់ ។

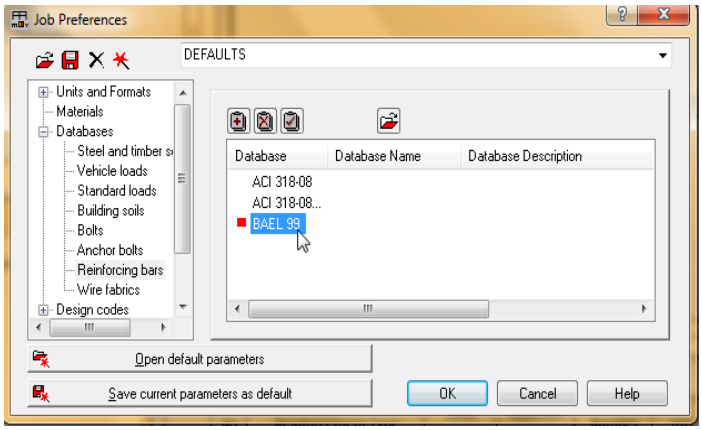


- រួចចុចលើ Tab Steel រួចត្រូវបំពេញ Name និង Description ដូចខាងក្រោម

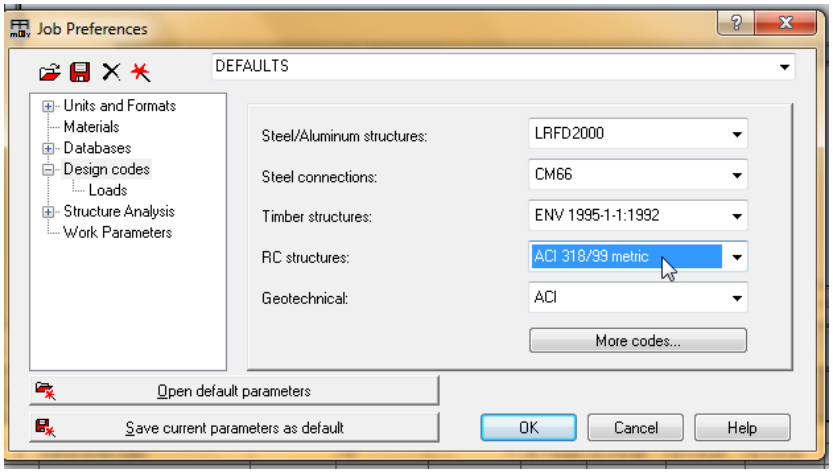


- រួចចុចលើ Tab Concrete រួចត្រូវបំពេញ Name និង Description និង Young modulus ដូចខាងក្រោម រួចចុច OK ។ រួចចុច Database > Reinforcing bars ដើម្បីជ្រើសរើសយក BAEL 99 រួចចុច ok ។

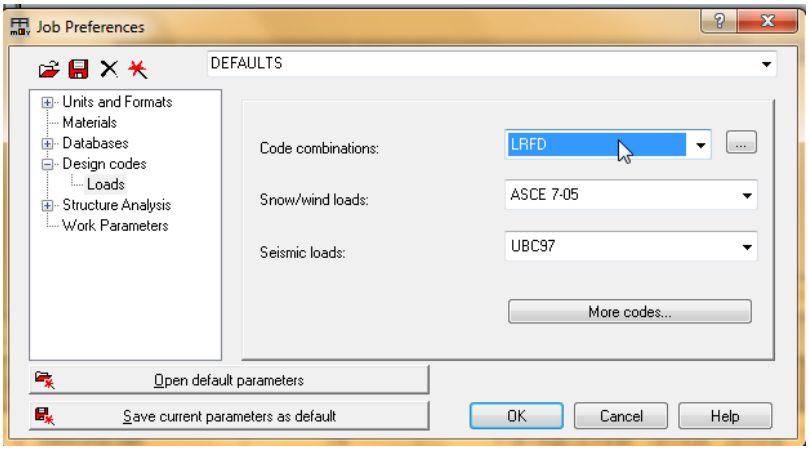




- រួចចុចលើ Design codes ក្នុងប្រអប់ RC Structures ត្រូវកំណត់ជា ACI 318/99 metric



- នៅត្រង់ Loads ក្នុងប្រអប់ Code combinations ត្រូវកំណត់ជា LRFD រួចចុច OK ។



បន្ទាប់មកចុច Save job Preference រួចចុច File name : ACI Code (Save as type : job Preference) រួចចុច Save , ok , Close . ។

១- **ការកំណត់** Beam - Definition **និង** Beam - Sections

១.១- កំណត់ធ្នឹមទី១ដែលមានប្រលោះ 8m

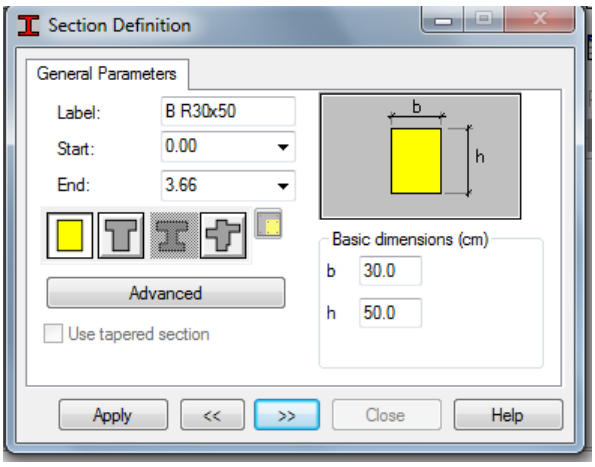
១.១.១- ការកំណត់មុខកាត់

1- Select លើធ្នឹមដែលចង់កំណត់មុខកាត់

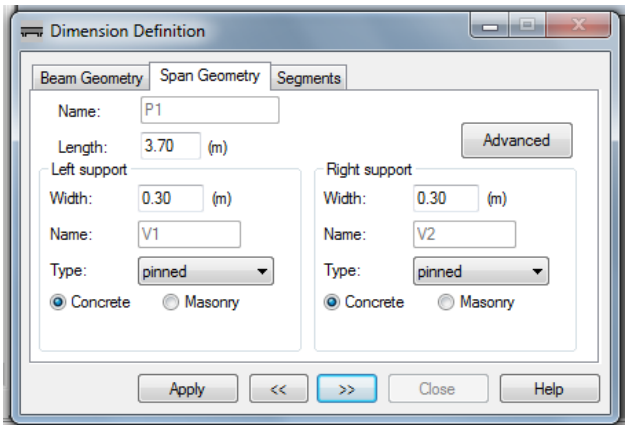
2- ចុចលើរូប  ដើម្បីកំណត់មុខកាត់ធ្នឹមរាងចតុកោណកែង

3- នៅក្នុងប្រអប់ Basic dimensions វាយតំលៃ b=30cm, h=50cm

4- រួចចុច Apply



១.១.២- ការកំណត់ប្រវែងរបស់ធ្នឹម និង ប្រភេទទំរ



1 -ចុចលើ Span Geometry

2 -នៅក្នុងប្រអប់ Length វាយតំលៃ 3.7m

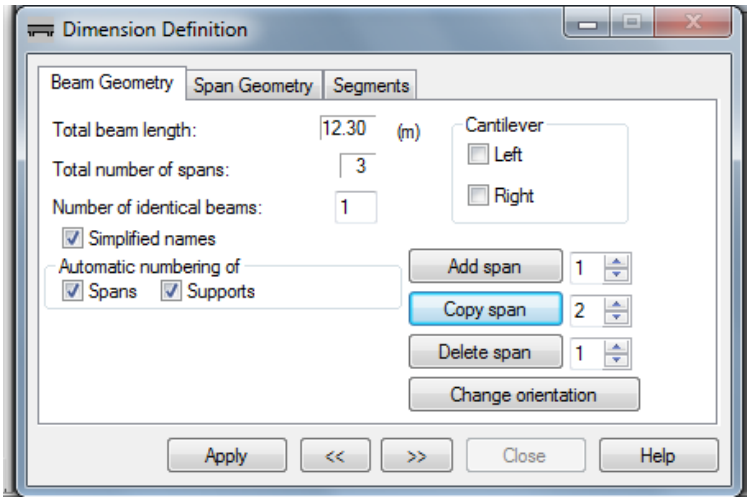
3 -នៅ Left Support វាយតំលៃនៅក្នុងប្រអប់ Width=0.3m សំរាប់មុខកាត់ទំរខាងឆ្វេង

4 -នៅ Right Support វាយតំលៃនៅក្នុងប្រអប់ Width=0.3m សំរាប់មុខកាត់ទំរខាងស្តាំ

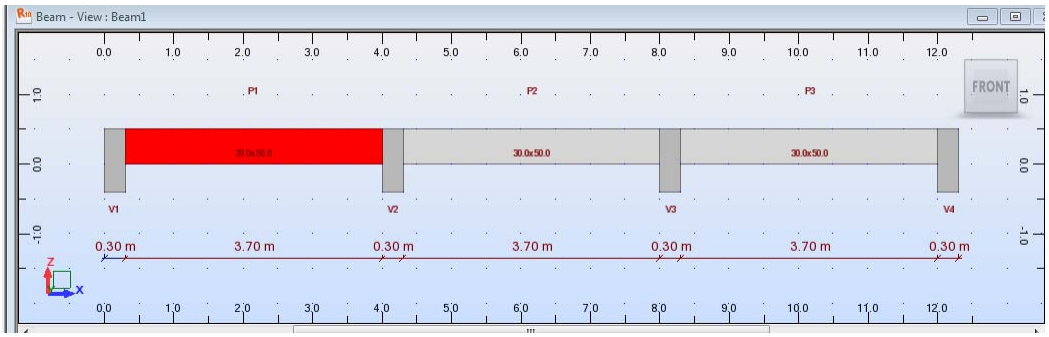
5- រួចចុច Apply

១.២- បន្ទាប់មកកំណត់ធ្នឹមទី២ និង ទី៣ ដែលមានប្រលោះ 8m និង 6m

- ដំបូងយើងធ្វើការ Copy ធ្នឹមទី១ ដែលបានកំណត់ហើយ ដោយបំពេញតំលៃ 2 នៅត្រង់ Copy Span នៅក្នុងប្រអប់ Beam Geometry រួចចុច Copy Span



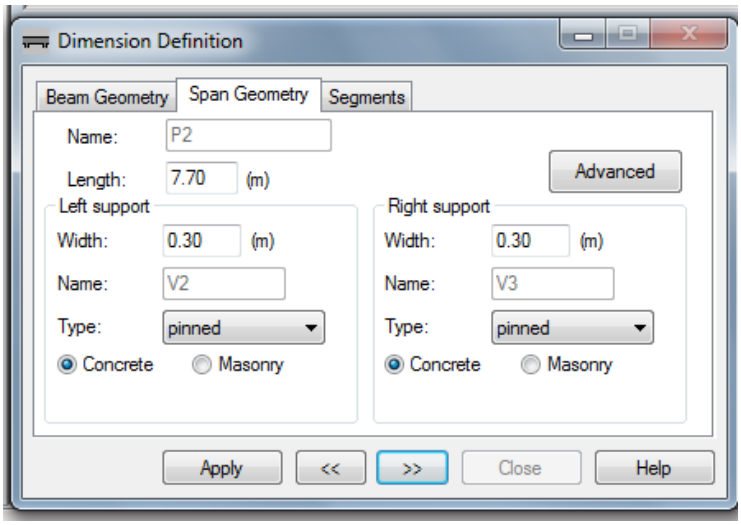
- រួចនិងទទួលបានរូបដូចខាងក្រោម



- ដោយសារប្រវែងប្រលោះដែលបាន Copy មិនដូចក្នុងប្រធាន ដូច្នេះយើងត្រូវកែតំរូវវាឡើងវិញដោយចុច Select ចុចលើធ្នឹមនីមួយៗ ជាមុនសិនទើបយើងកែ ។ ប៉ុន្តែយើងកែតែប្រវែងប្រលោះរបស់ធ្នឹមប៉ុណ្ណោះ ពីព្រោះធ្នឹមទាំងអស់មានមុខកាត់ដូចគ្នា ។

១.៣- ការកែតំរូវប្រវែងប្រលោះរបស់ធ្នឹម

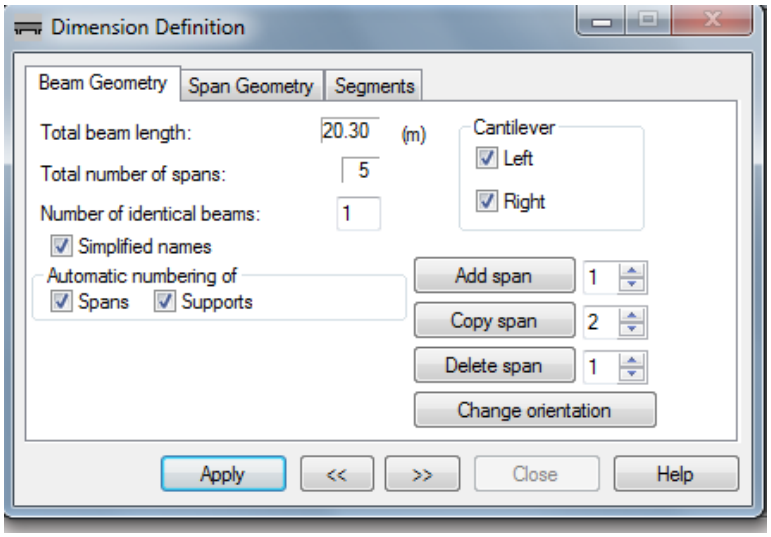
- ចុចលើធ្នឹមទី ២ គឺឈ្មោះ P2 បន្ទាប់មកធ្វើការកែប្រវែងប្រលោះរបស់ធ្នឹម នៅក្នុងផ្ទាំង Dimension Definition ត្រង់ Span Geometry ក្នុងប្រលោះ Length វាយលេខ 7.7m ចូលដូចរូបរួច Apply



- រីឯធ្នឹមទី ៣ គឺឈ្មោះ P3 យើងអនុវត្តន៍ដូចទៅនឹងធ្នឹមទី ២ ដែលបានកែរួចហើយនៅខាងលើ ។

១.៤- ធ្វើការកំណត់ធ្នឹម Cantilever ឆ្វេង (1.5m) និង ស្តាំ (1.5m)

- ដោយចុចលើផ្ទាំង Dimension Definition ត្រង់ Beam Geometry រួច Tick យក Left និង Right ត្រង់ Cantilever រួចចុច Apply



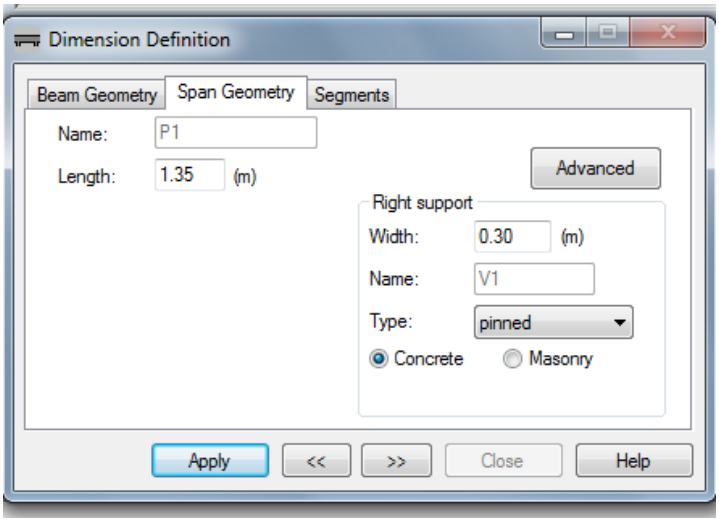
- រួចកំណត់ប្រវែង Cantilever

1 - Select Cantilever ឆ្វេងដោយប្រើ Mouse

វាយតំលៃ 1.35m នៅក្នុងប្រអប់ Length រួចចុច Apply

2 - Select Cantilever ស្តាំ ដោយប្រើ Mouse

វាយតំលៃ 1.35m នៅក្នុងប្រអប់ Length រួចចុច Apply



២ - ការកំណត់ប្រហោងរបស់ធ្នឹមដើម្បីដាក់សំរាប់ដាក់ខ្សែភ្លើង ឬ ទុយោ Beam – Opening

ការកំណត់ធ្នឹមរងនៅប្រលោះ 8m ធ្នឹមទី ៣ គឺ P3

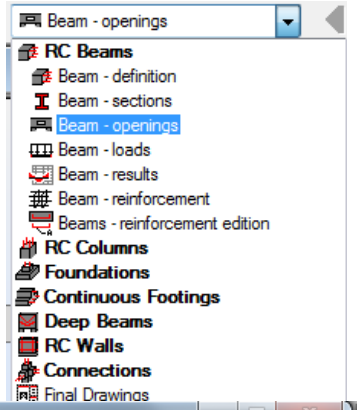
- មុនដំបូងត្រូវ Select ធ្នឹម P3 សិនរួចចុច Icon  នេះនៅស្តាំអេក្រង់ រឺ ចុចលើ Layout

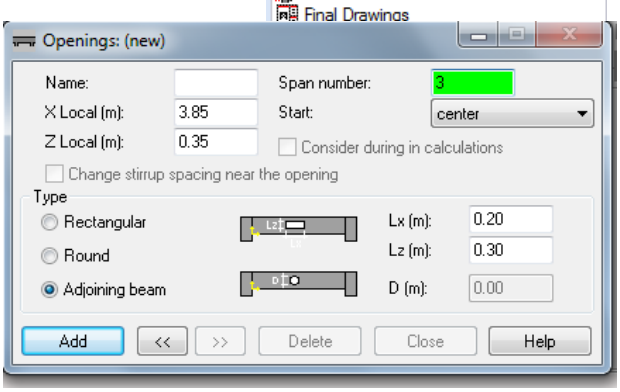
1- ក្នុងប្រអប់ Span number វាយលេខ 3
មានន័យថាដាក់ធ្នឹមរងត្រង់ធ្នឹមទី ៣

2- ក្នុងប្រអប់ X Load (m) វាយ 3.85m
និងក្នុងប្រអប់ Y Load (m) វាយ 0.35m

3- ត្រង់ Type Tick យក Adjoining beam
ហើយក្នុងប្រអប់ Lx (m) វាយ 0.2m
និងក្នុងប្រអប់ Lz (m) វាយ 0.3m
មានន័យថា b=20cm និង h=30cm

4- រួចចុច Add





៣- ការកំណត់ការដាក់បន្ទុកលើធ្នឹម Beam – Loads

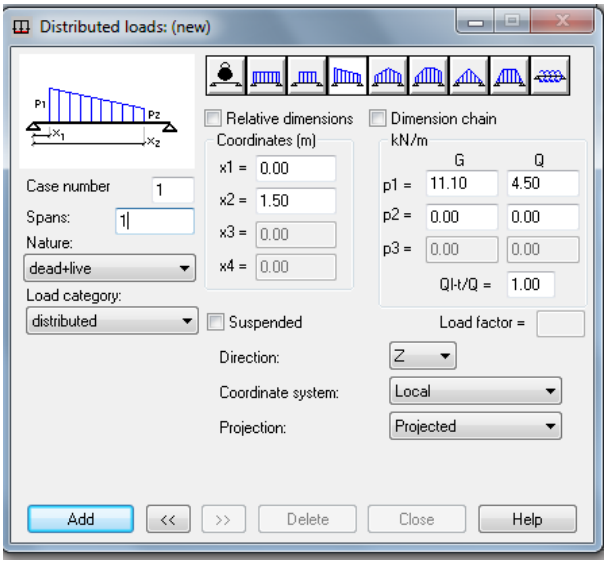
- ធ្នឹម P1 : DL=3.7kN/m^2 x 1.5m x 2 = 11.1kN/m
  LL=1.5kN/m^2 x 1.5m x 2 = 4.5kN/m
- ធ្នឹម P2 : DL=3.7kN/m^2 x 2m x 2 = 14.8kN/m
  LL=1.5kN/m^2 x 2m x 2 = 6kN/m
- ធ្នឹម P3 : DL=3.7kN/m^2 x 2m x 2 = 14.8kN/m
  LL=1.5kN/m^2 x 2m x 2 = 6kN/m
- ធ្នឹម P4 : DL=3.7kN/m^2 x 2m x 2 = 14.8N/m
  LL=1.5kN/m^2 x 2m x 2 = 6kN/m
- ធ្នឹម P5 : DL=3.7kN/m^2 x 1.5m x 2 = 11.1kN/m
  LL=1.5kN/m^2 x 1.5m x 2 = 4.5kN/m
- ការដាក់បន្ទុកលើធ្នឹមរង DL=25kN/m^3 x 4m x 0.2m x 0.3m = 6kN
  LL=1.5kN/m^2 x 4m x 0.2m = 1.2kN



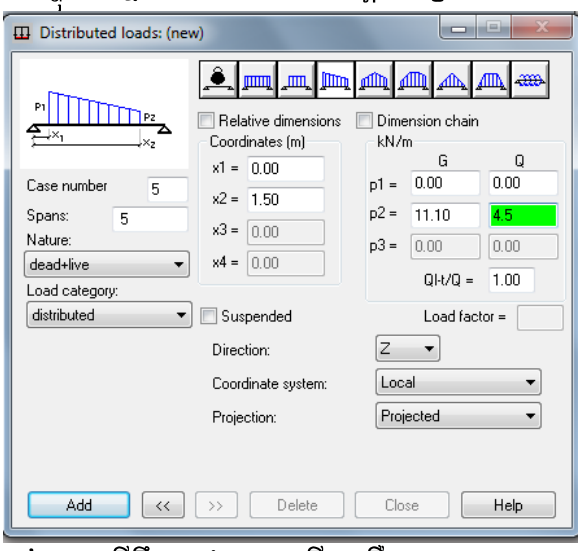
៣.១- ការដាក់បន្ទុកលើធ្នឹម :

- ដំបូងចុចលើ Icon  នេះនៅស្តាំអេក្រង់ រឺចុចលើ Layout

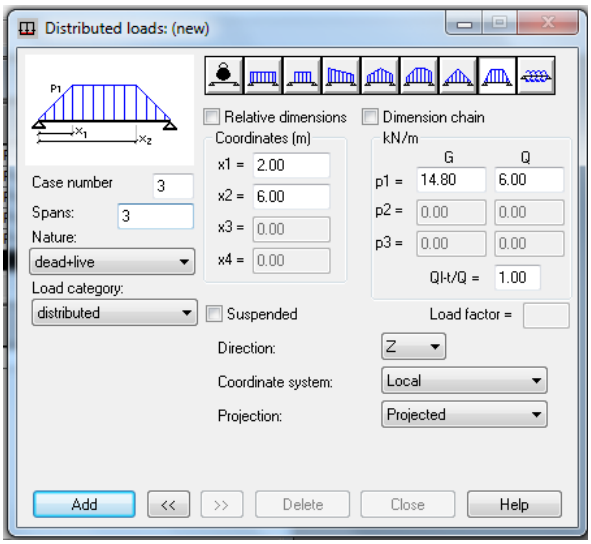
៣.១.១-ការដាក់បន្ទុកលើធ្នឹម Cantilever ឆ្វេងនៅប្រលោះទី១ គឺ P1 រួចចុច Add



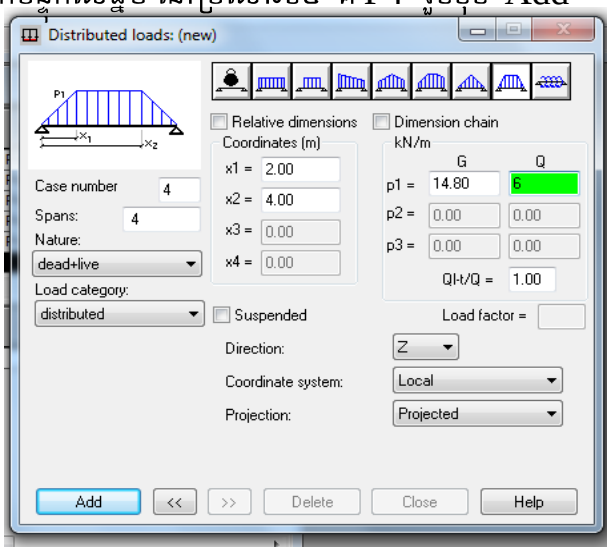
៣.១.២-ការដាក់បន្ទុកលើធ្នឹម Cantilever ស្តាំនៅប្រលោះទី៥ គឺ P5 រួចចុច Add



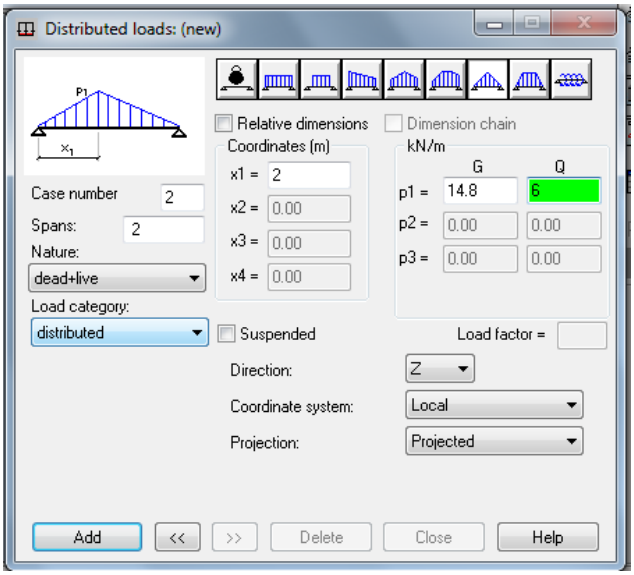
៣.១.៣-ការដាក់បន្ទុកលើធ្នឹម នៅប្រលោះទី៣ គឺ P3 រួចចុច Add



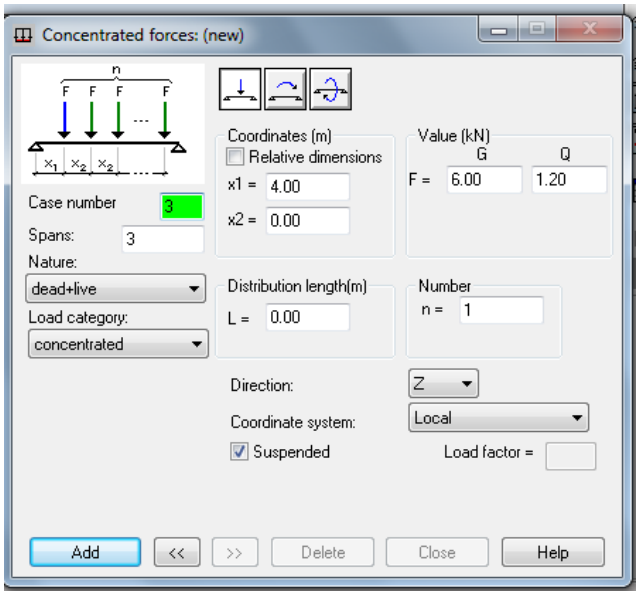
៣.១.៤-ការដាក់បន្ទុកលើធ្នឹម នៅប្រលោះទី៤ គឺ P4 រួចចុច Add



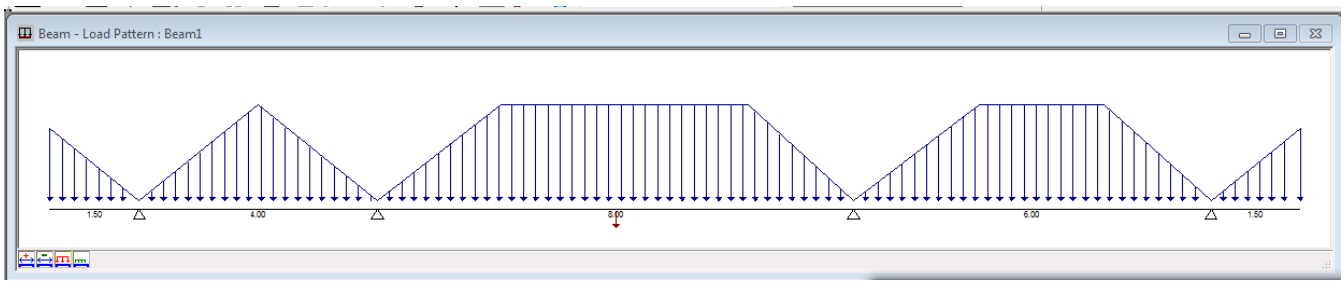
៣.១.៥-ការដាក់បន្ទុកលើធ្នឹម នៅប្រលោះទី២ គឺ P2 រួចចុច Add



៣.១.៥-ការដាក់បន្ទុកលើធ្នឹមរង នៅប្រលោះទី៣ គឺ P3 រួចចុច Tick នៅត្រង់ Suspended រួចចុច Add

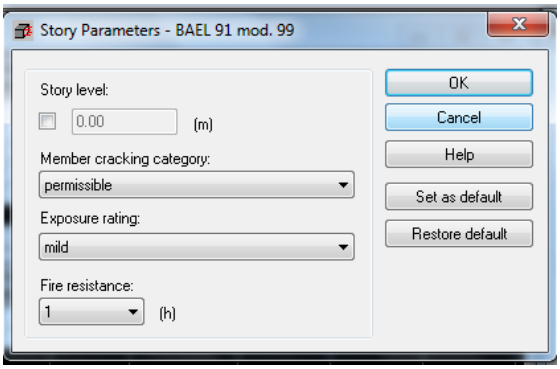


- នៅពេលដាក់បន្ទុករួចហើយអេក្រង់នឹងបង្ហាញដូចរូបខាងក្រោម :



៤- ការកំណត់ Story Parameters

- ដំបូងចុចលើ Icon  នេះនៅស្តាំអេក្រង់ រឺចុចលើ Analysis Menu > Story Parameters រួចចុច OK

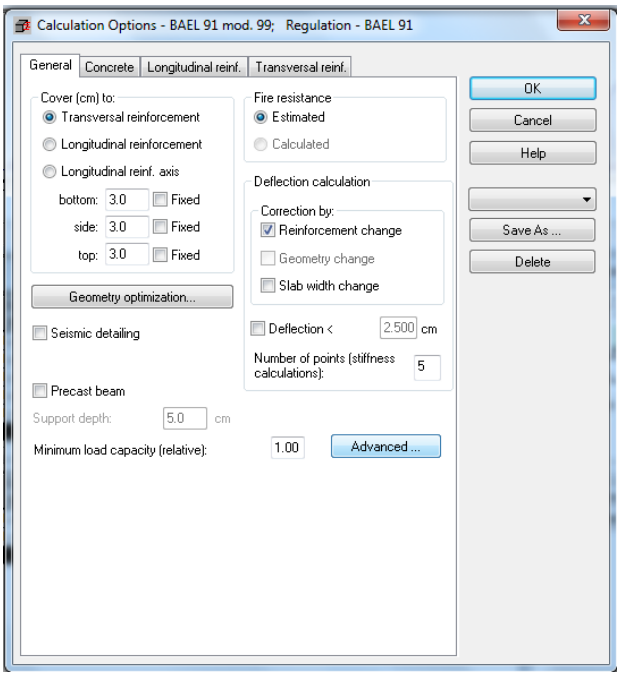


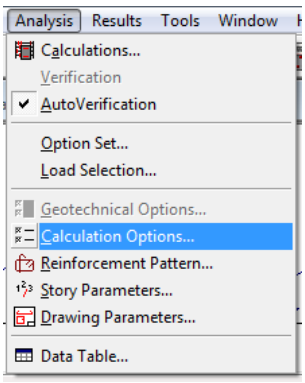
៥- ការកំណត់ Calculation Options

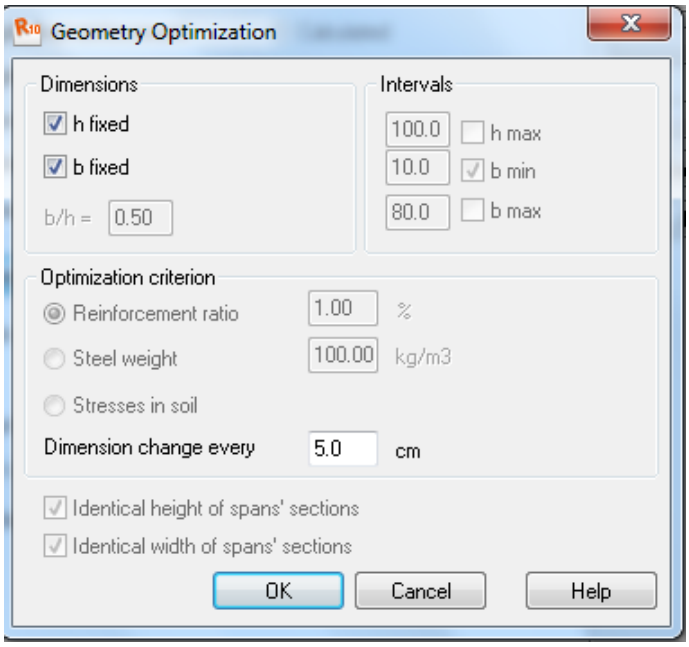
នៅក្នុង Calculation Options មាន ៤ Tab ដែលត្រូវកំណត់ :

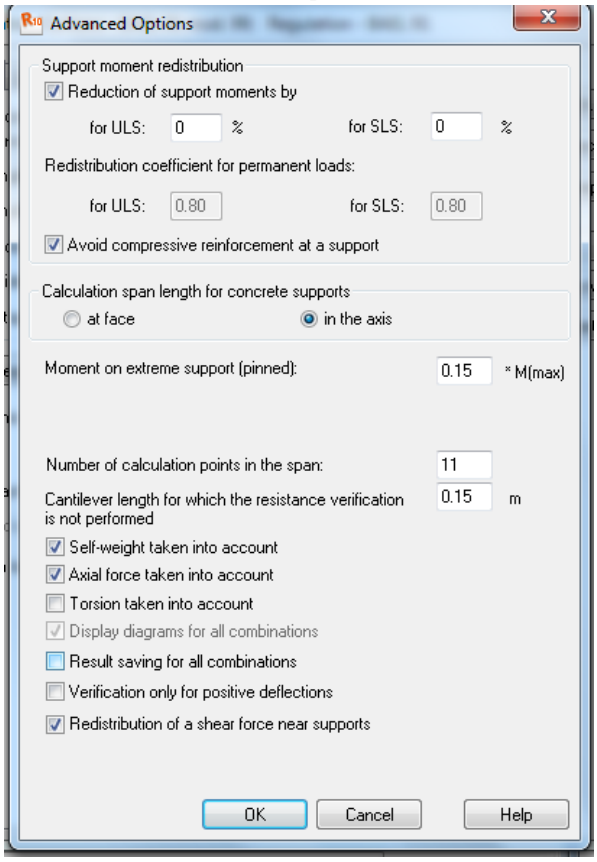
- Tab General : កំណត់ស្រទាប់ការពារ ផ្ទៀងផ្ទាត់ភាពដាប និង ការកំណត់ផ្សេងទៀត.....
- Tab Concrete : កំណត់រេស៊ីស្តង់សង្កត់របស់បេតុងនៅអាយុ ២៨ ថ្ងៃ
- Tab Longitudinal reinf.. កំណត់រេស៊ីស្តង់ដែកបណ្តោយ និងអង្កត់ផ្ចិត
- Tab Traversal reinf.. កំណត់រេស៊ីស្តង់ដែកកង និងអង្កត់ផ្ចិត
- ដំបូងចុចលើ Icon  នេះនៅស្តាំអេក្រង់ រឺចុចលើ Analysis Menu > Calculation Options រួចចុច OK

Tab General :

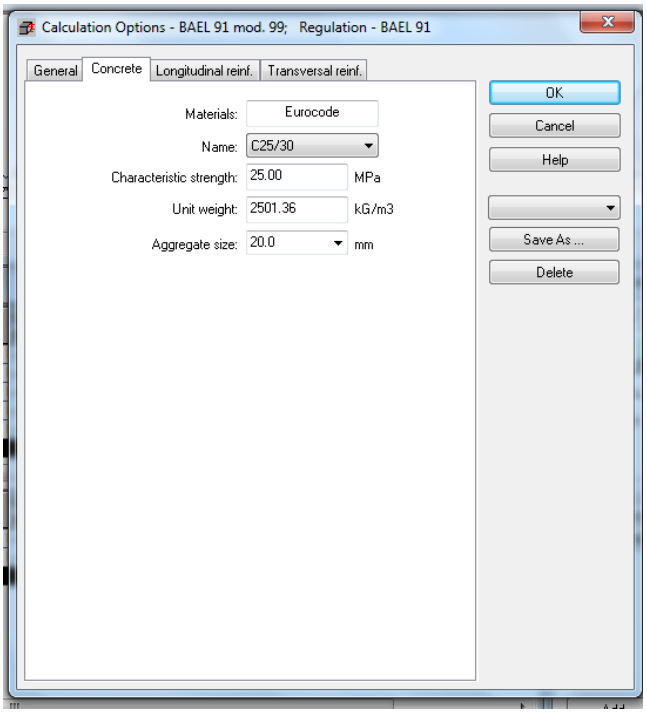




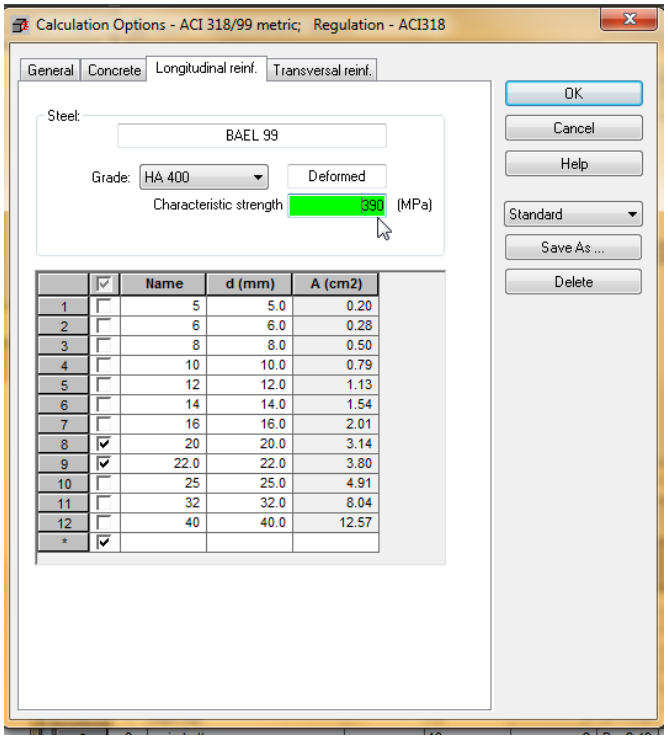




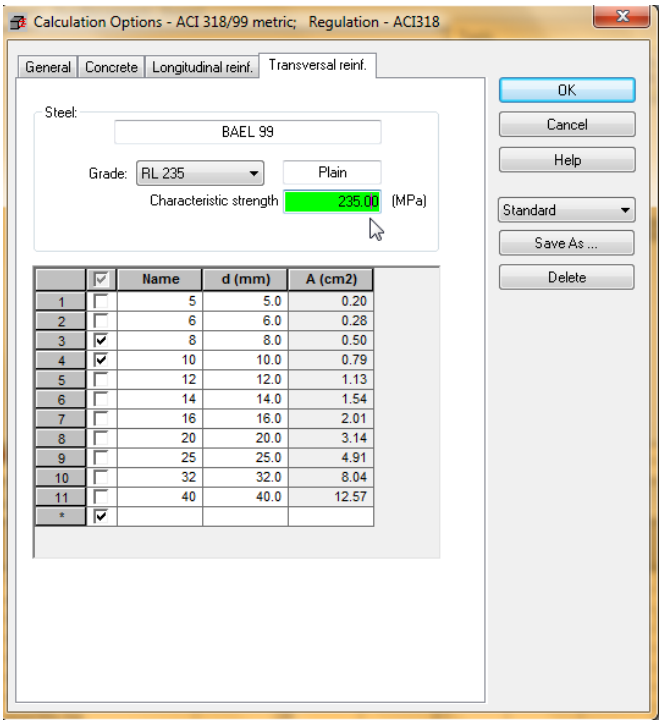
Tab Concrete :



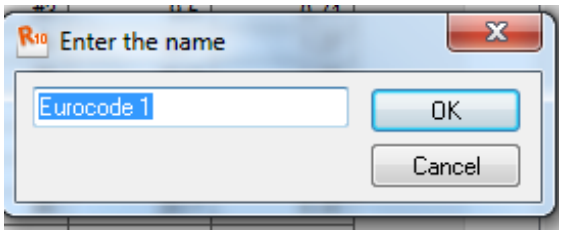
Tab Longitudinal reinf…:



Tab Traversal reinf..



- នៅពេលកំណត់រួចរាល់ហើយ ចុចលើ Save As ….. វាយឈ្មោះ Euro code 1 រួចចុច ok ដើម្បីរក្សាទុក ។

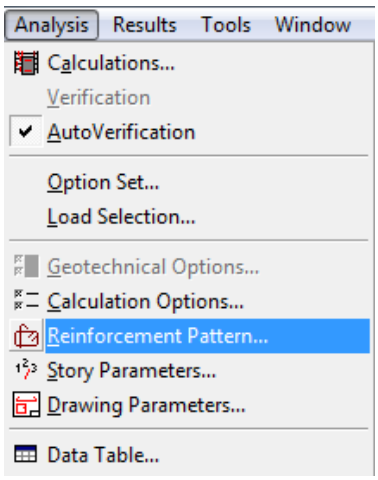


៦ - ការកំណត់ Reinforcement Pattern

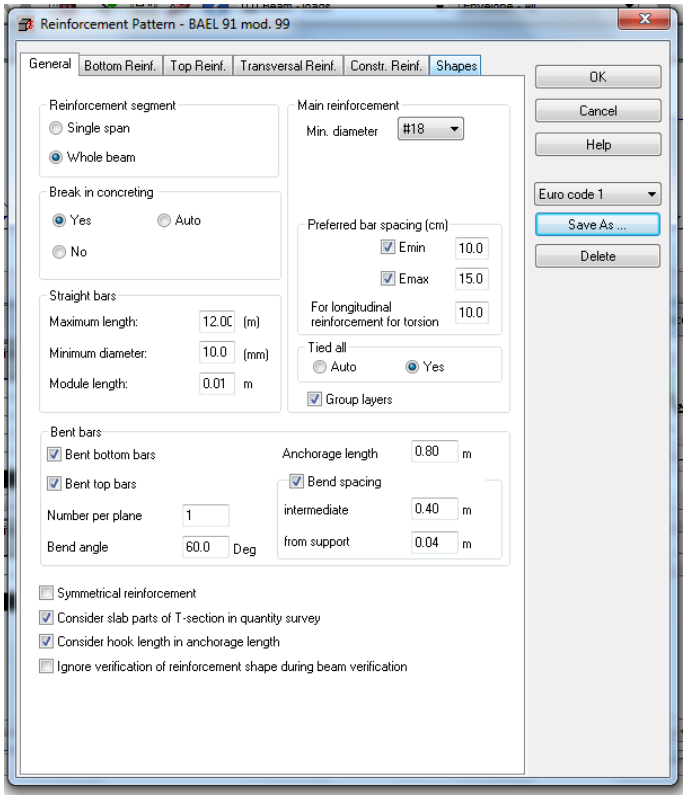
នៅក្នុង Reinforcement Pattern មាន ៦ Tab ដែលត្រូវកំណត់ :

- Tab General : កំណត់ការដាក់ដែកកង Segment គំលាតពីសសៃដែក១ ទៅសសៃដែក ១.....
- Tab Bottom Reinf : កំណត់ការដាក់សសៃដែកបណ្តោយខាងក្រោមធ្នឹម និងការបង្កប់ដែក
- Tab Top Reinf : កំណត់ការដាក់សសៃដែកបណ្តោយខាងលើធ្នឹម
- Tab Transversal Reinf : កំណត់ការដាក់ដែកកងទាំងនៅកណ្តាលប្រលោះ ត្រង់ទំរ និងធ្នឹមរង
- Tab Constr Reinf : កំណត់ការដាក់សសៃដែកបណ្តោយដែលមិនធ្វើការ និងការតដែក
- Tab Shapes : កំណត់ទ្រង់ទ្រាយនៃការពត់ដែក
-

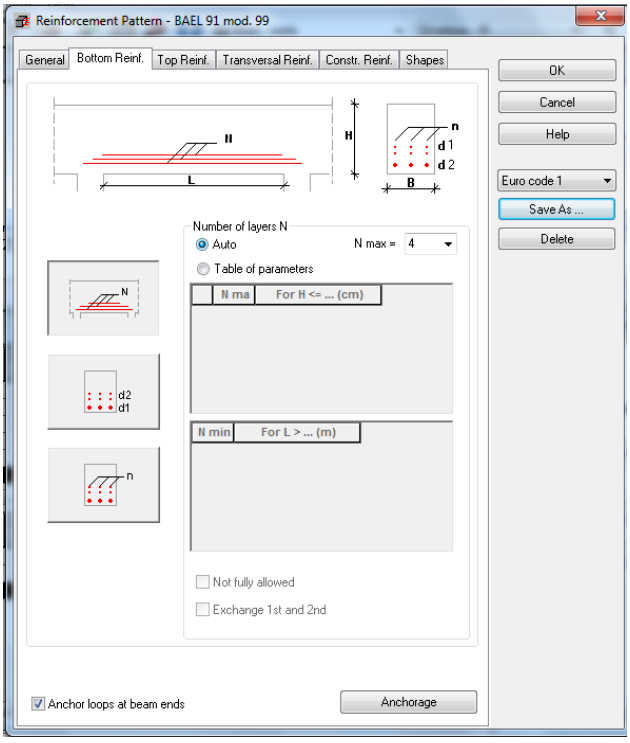
- ដំបូងចុចលើ Icon  នេះនៅស្តាំអេក្រង់ រួចចុចលើ Analysis Menu > Reinforcement Options

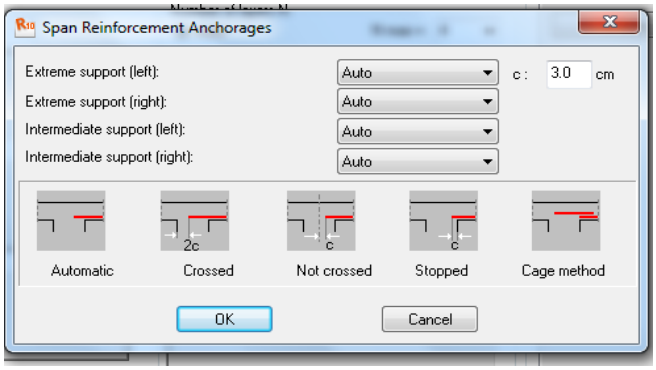


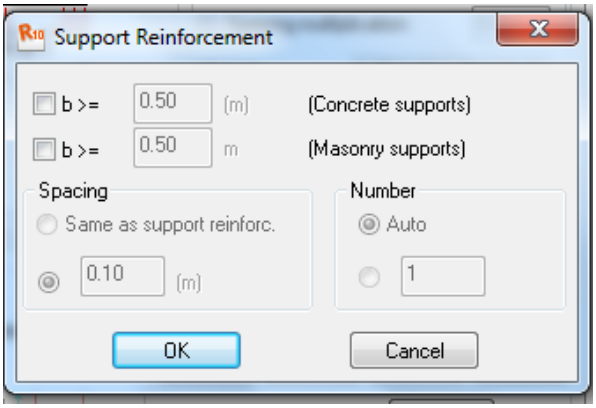
Tab General :

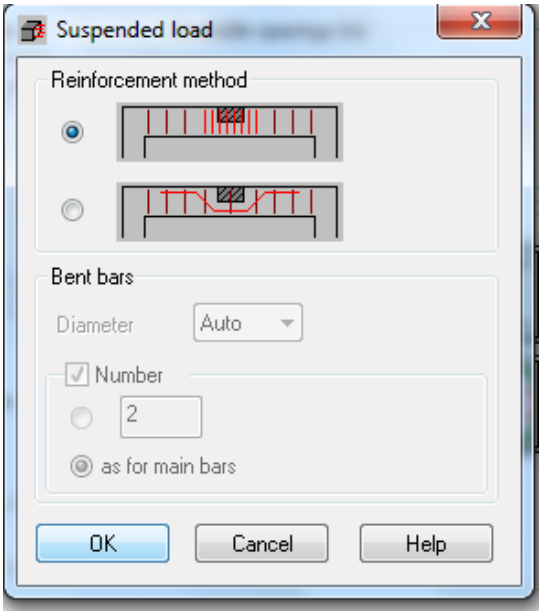


Tab Bottom Reinf :

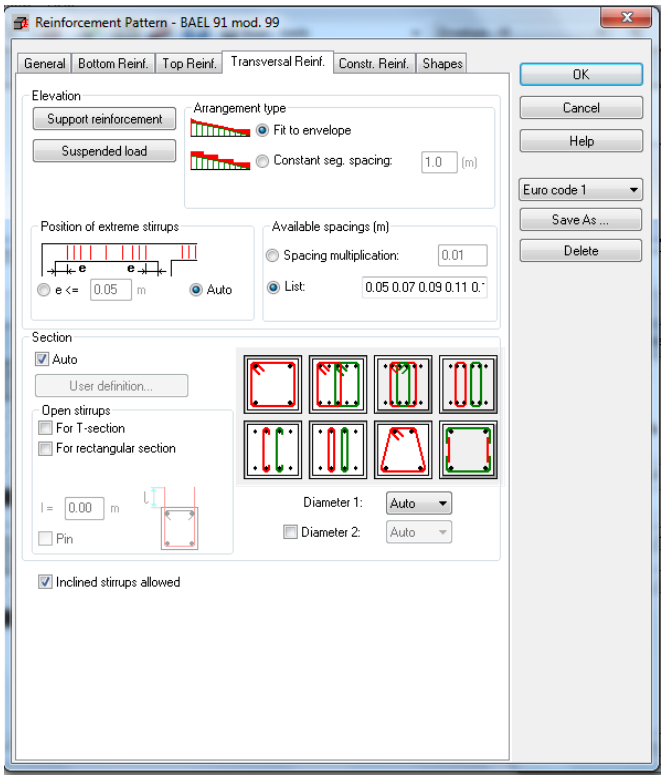




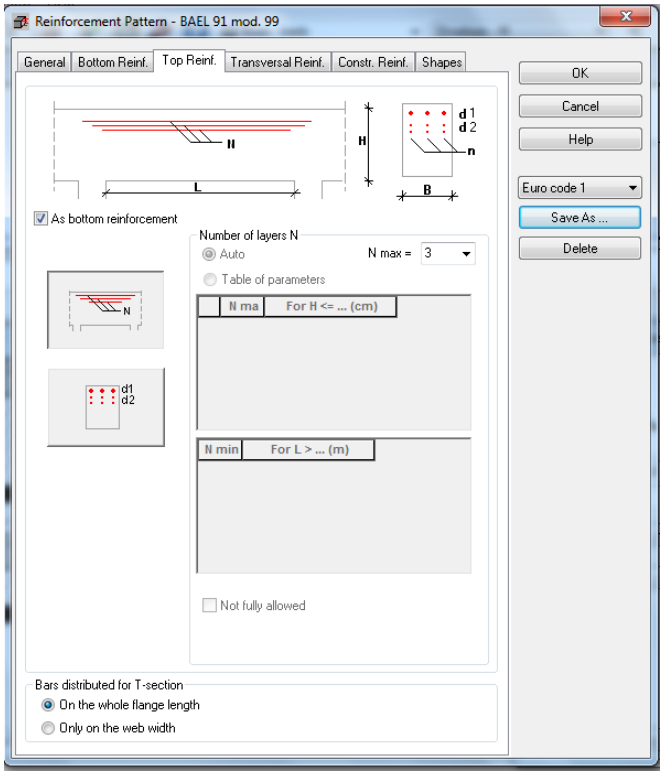




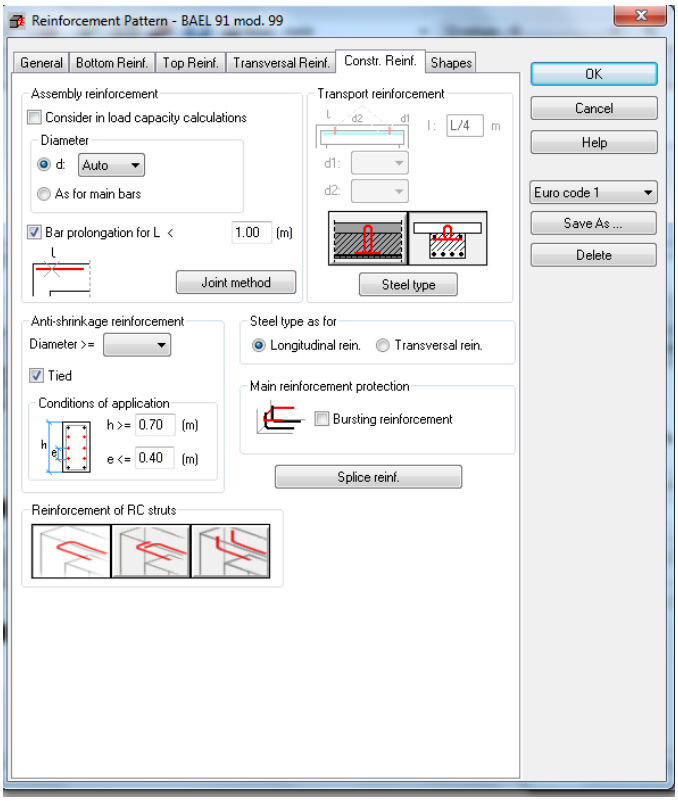
Tab Transversal Reinf :

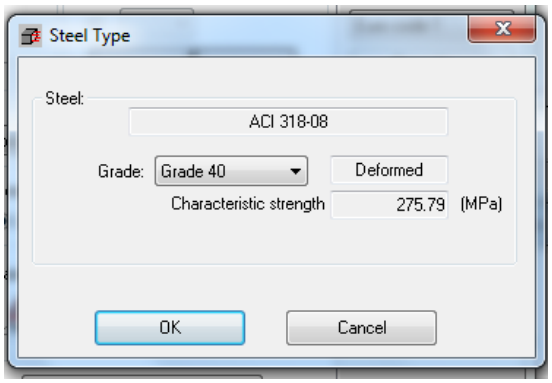


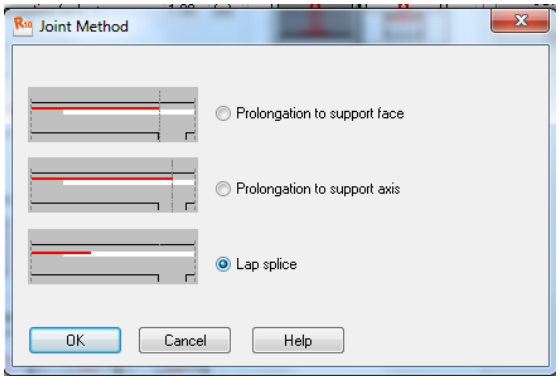
Tab Top Reinf :

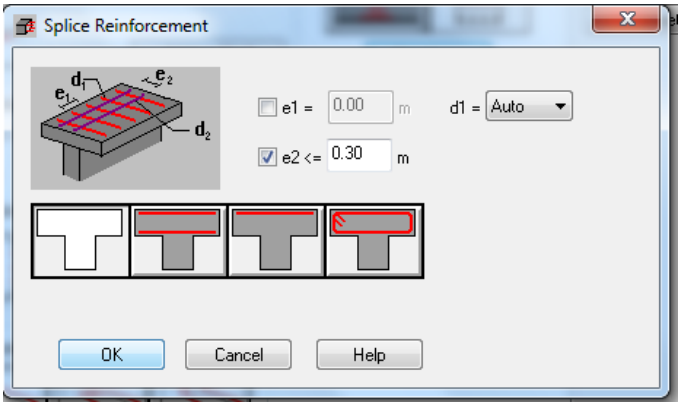


Tab Constr Reinf :

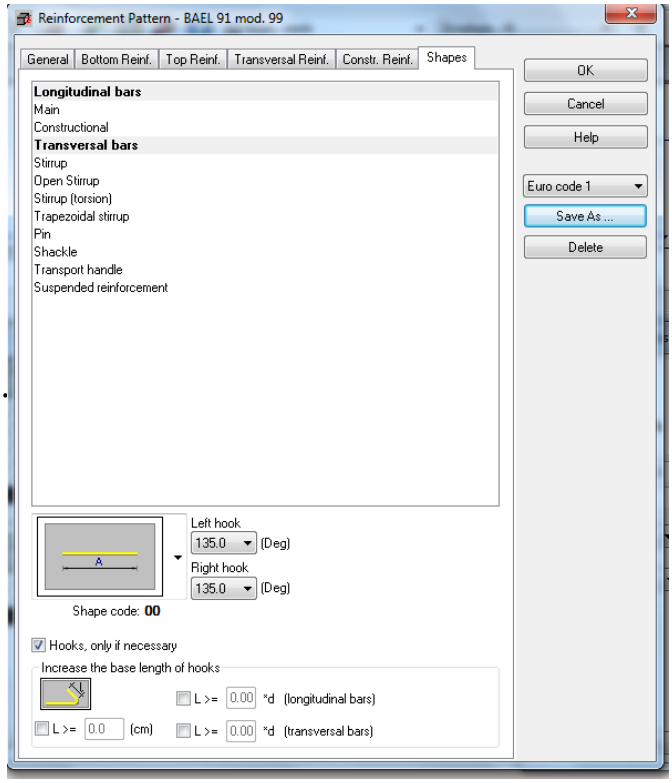




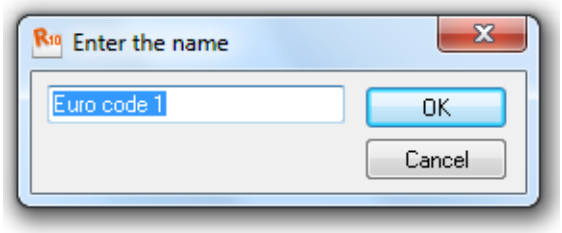




Tab Shapes :

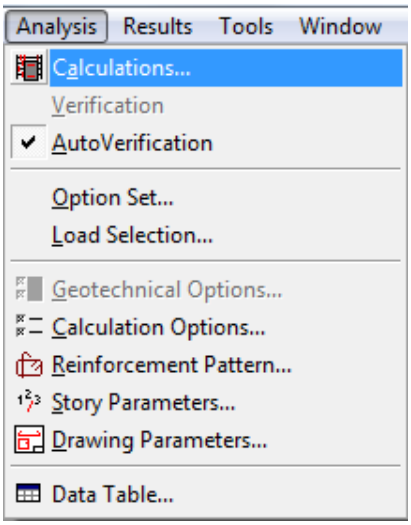


- នៅពេលកំណត់រួចរាល់ហើយចុចលើ Save As ..... វាយឈ្មោះ Euro code 1 រួចចុច ok ដើម្បីរក្សាទុក ។

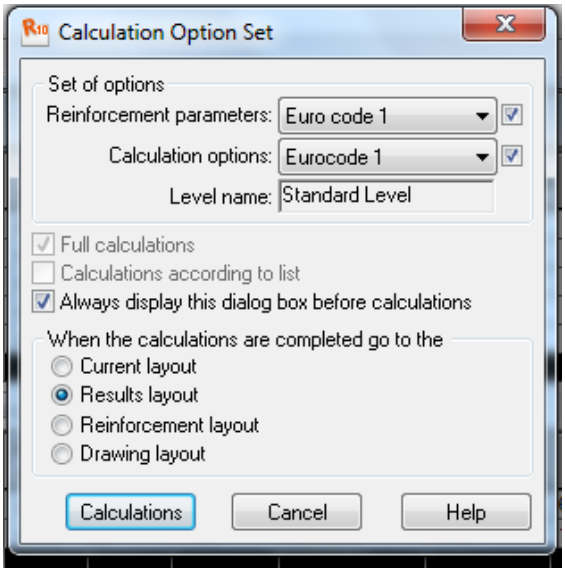


៧-ចាប់ផ្តើមគណនារកសសៃវែក Start Calculate

- ដំបូងចុចលើ Icon  នេះនៅស្តាំអេក្រង់រឺចុចលើ Analysis Menu > Calculations……



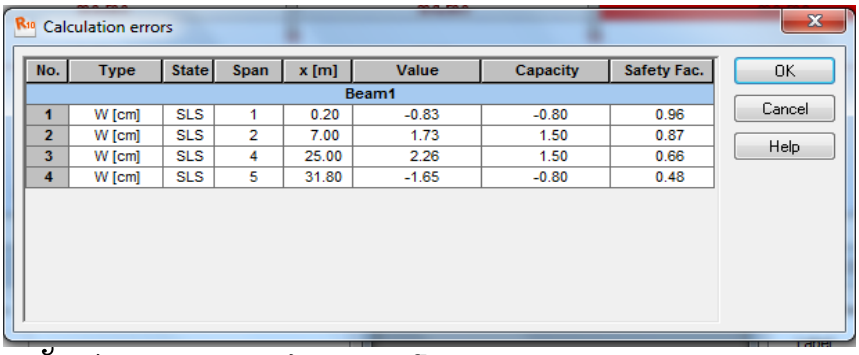
ពេលនោះវានឹងបង្ហាញផ្ទាំង Calculation Option Set ដូចរូប



ជ្រើសរើសឈ្មោះត្រង់ Reinforcement parameters និង Calculation options ដែលយើងបានកំណត់រួចមក ហើយនៅចំនុចទី ៥ និង ទី៦ ខាងលើគឺឈ្មោះថា Euro code ។

-ករណីបើមាន Error & Warning វានឹងបង្ហាញផ្ទាំង Calculation Errors បន្ទាប់ពីចុច Calculation

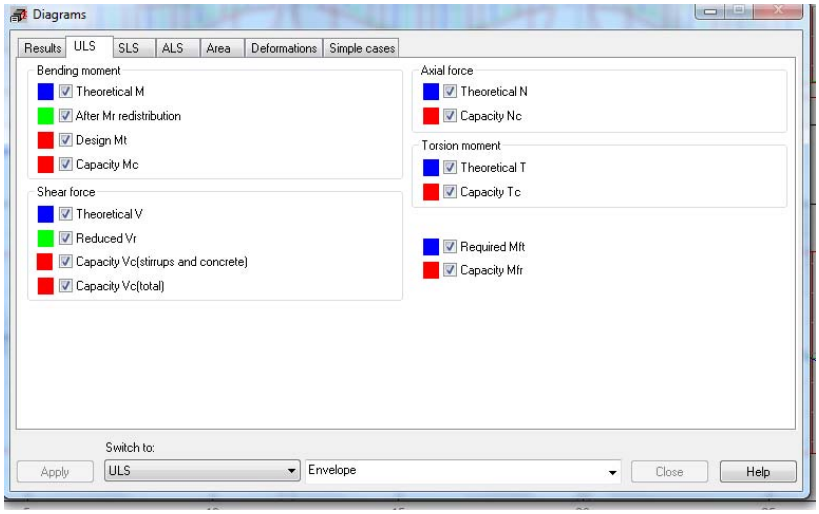
-ករណីបើគ្មាន Error & Warning ទេវានឹងមិនបង្ហាញផ្ទាំង Calculation Errors



No.	Type	State	Span	x [m]	Value	Capacity	Safety Fac.
Beam1							
1	W [cm]	SLS	1	0.20	-0.83	-0.80	0.96
2	W [cm]	SLS	2	7.00	1.73	1.50	0.87
3	W [cm]	SLS	4	25.00	2.26	1.50	0.66
4	W [cm]	SLS	5	31.80	-1.65	-0.80	0.48

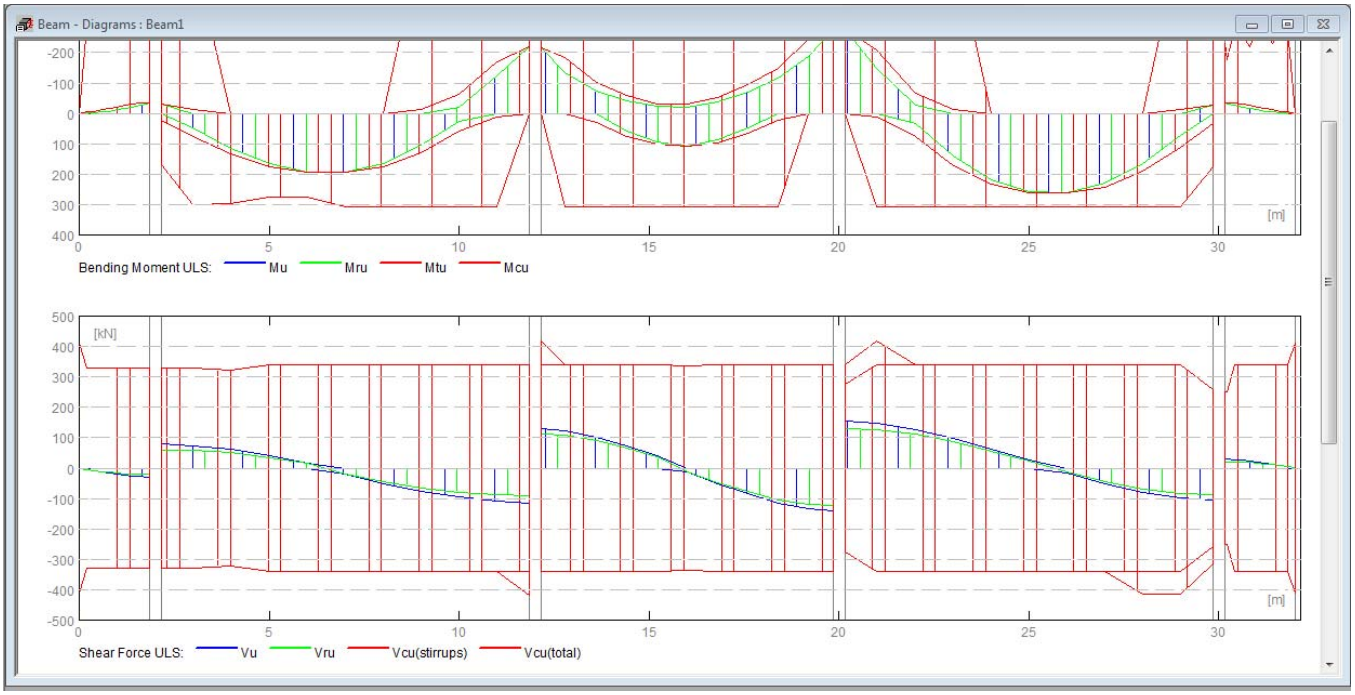
- W មានន័យថា Error ភាពដាប Deflection
- M មានន័យថា Error ម៉ូម៉ង់ Moment

- V មានន័យថា Error កំលាំងកាត់ទទឹង Shear force
- នៅពេល Program ប្រាប់ថា Error ត្រង់ណាហើយ យើងចុច ok ដើម្បីបិទផ្ទាំង Calculation Error បន្ទាប់មកត្រូវចុចទៅលើ Tab ULS ក្នុងតារាងខាងក្រោមដើម្បីមើលដ្យាក្រាម ម៉ូម៉ង់ កំលាំងកាត់ទទឹង និង ភាពដាប ដូចក្នុងរូបខាងក្រោម ។

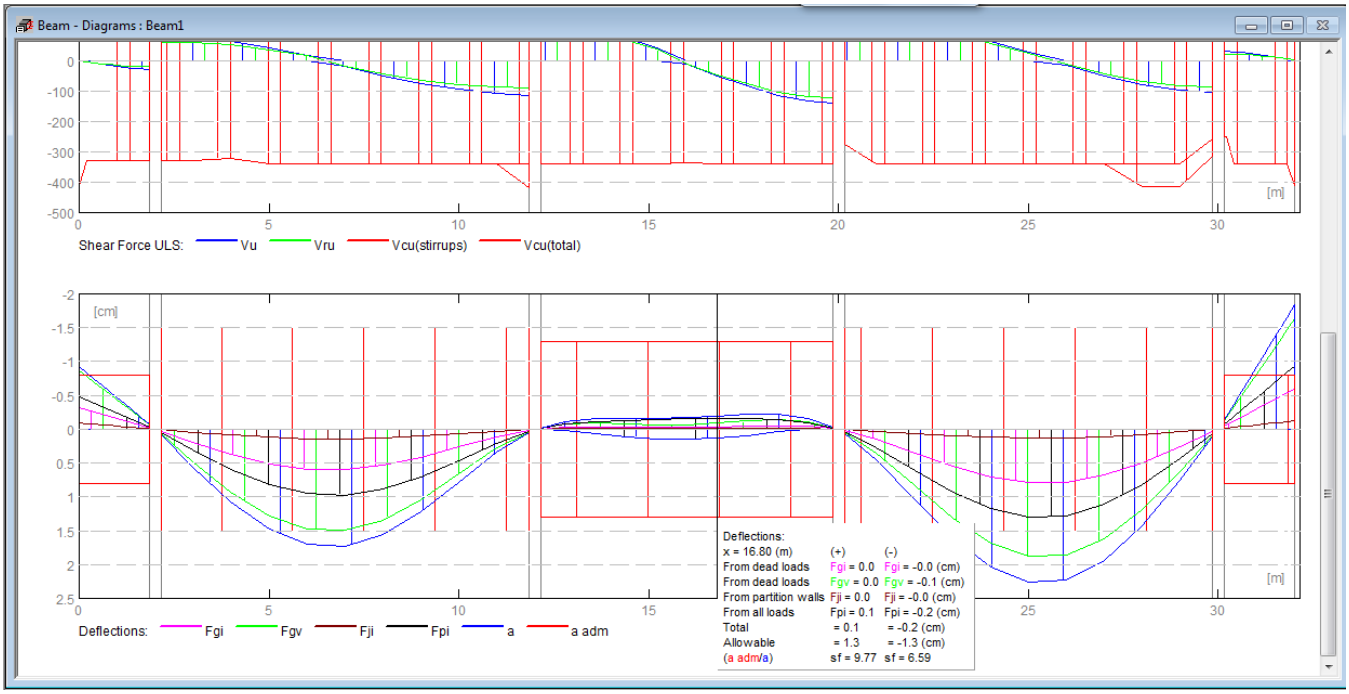


ដ្យាក្រាម ម៉ូម៉ង់ និង កំលាំងកាត់ទទឹង

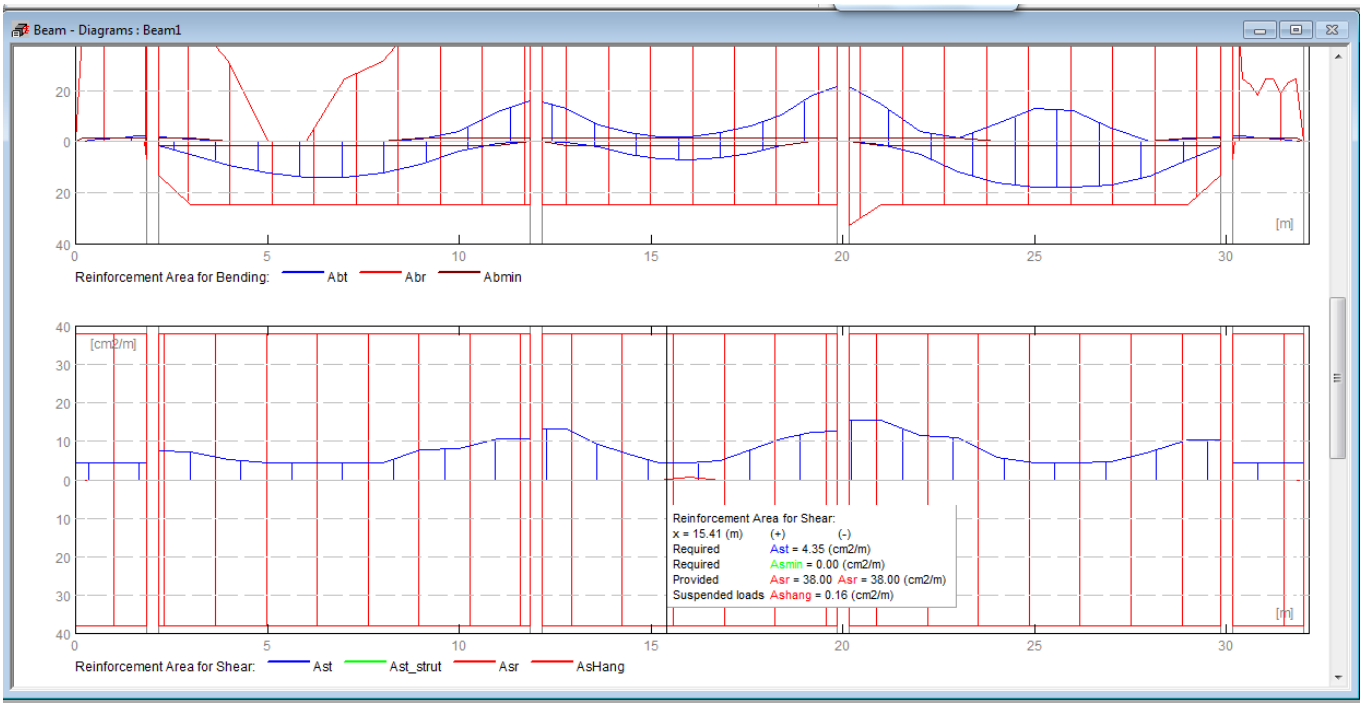
Moment and Shear force Diagram



ដ្យាក្រាម ភាពដាប
Deflection Diagram



ដ្យាក្រាម ដែក
Reinforcement Diagram



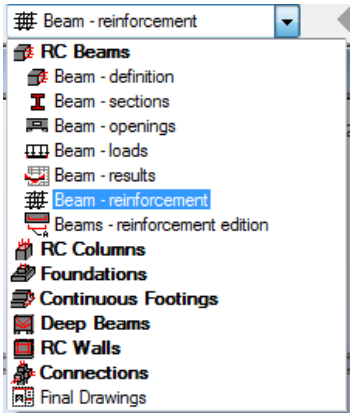
1- ULS : Diagram of Ultimate Limit State

www.yerngshare.blogspot.com

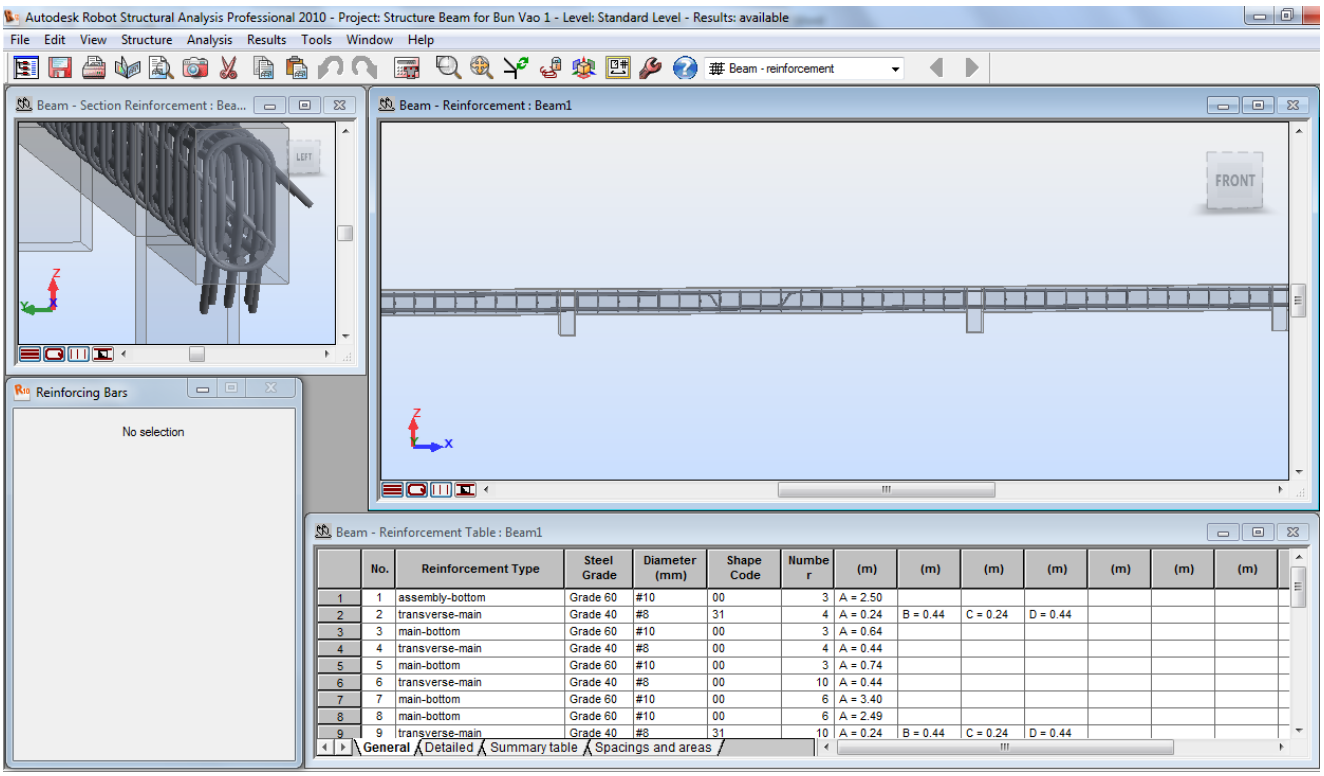
2- SLS : Diagram of Serviced Limit State
3- ALS : Diagram of Accidental Limit State
4- Area : Diagram of Longitudinal and Transversal Reinforcement
5- Deformations : Diagram of Beam Deflection
6- Simple Cases : Diagram of Simple Cases

៨- <u>ការត្រួតពិនិត្យមើលការដាក់រ៉ៀបចំសែសែក</u> <u>Beam – Reinforcement</u>

- ដំបូងចុចលើ Icon  នេះនៅស្តាំអេក្រង់រឺ ចុចលើ Layout ដូចរូប

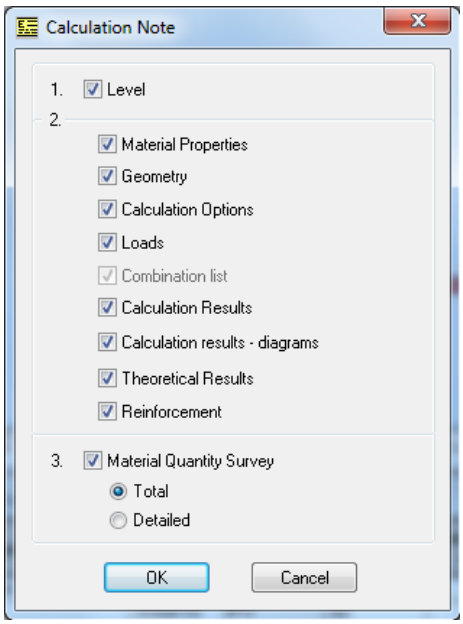


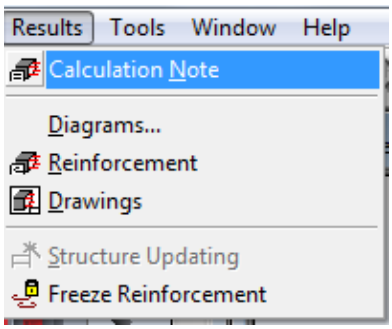
បង្ហាញដូចរូប :



៩- កំណត់សំគាល់ការគណនា Calculation Note

- ដំបូងចុចលើ Icon  នេះនៅស្តាំអេក្រង់រឺ ចុចលើ Results Menu > Calculation Note

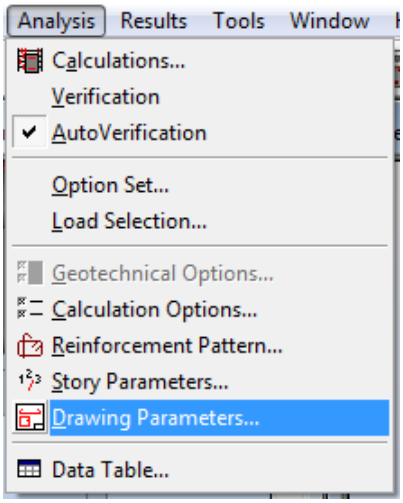


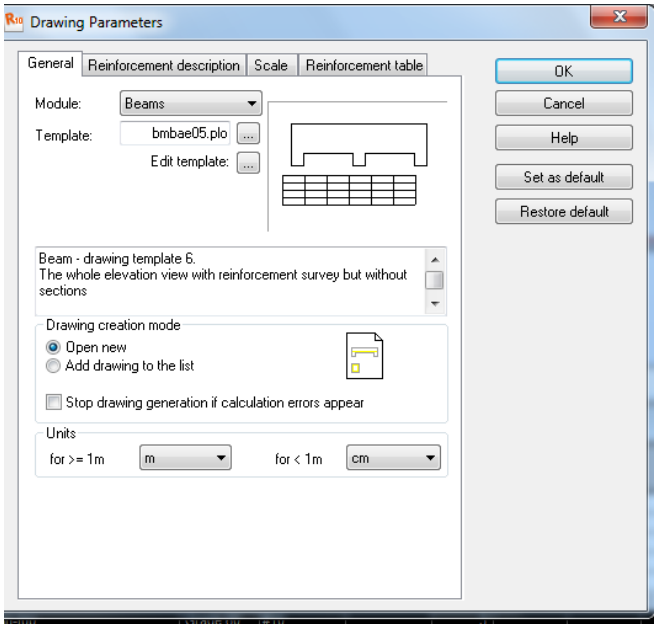


១០- Drawing Parameters

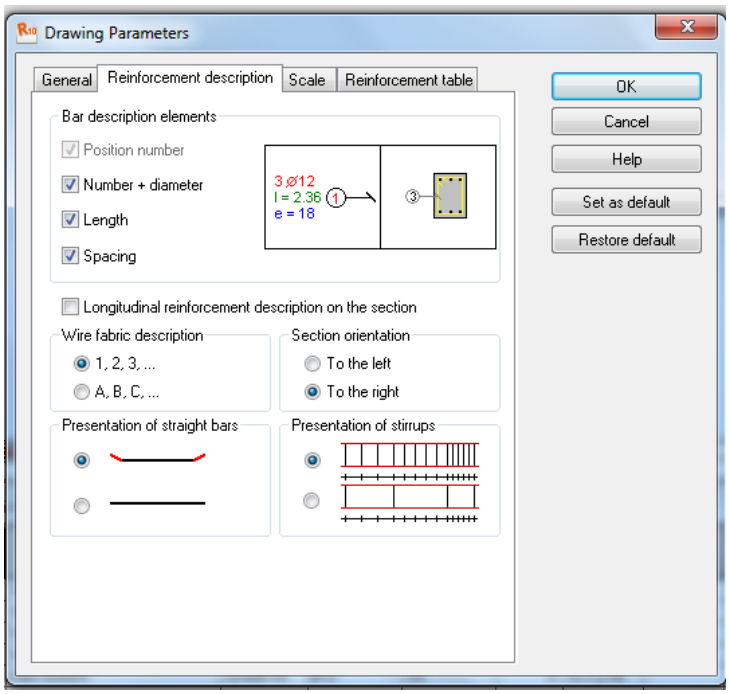
- ដំបូងចុចលើ Icon  នេះនៅស្តាំអេក្រង់រឺ ចុចលើ Analysis > Drawing Parameter

Tab General : ត្រង់កន្លែង Template ជ្រើសរើសយក ពាក្យថា bmbae05.plo សំរាប់មើល Drawing ទាំងមូល ។

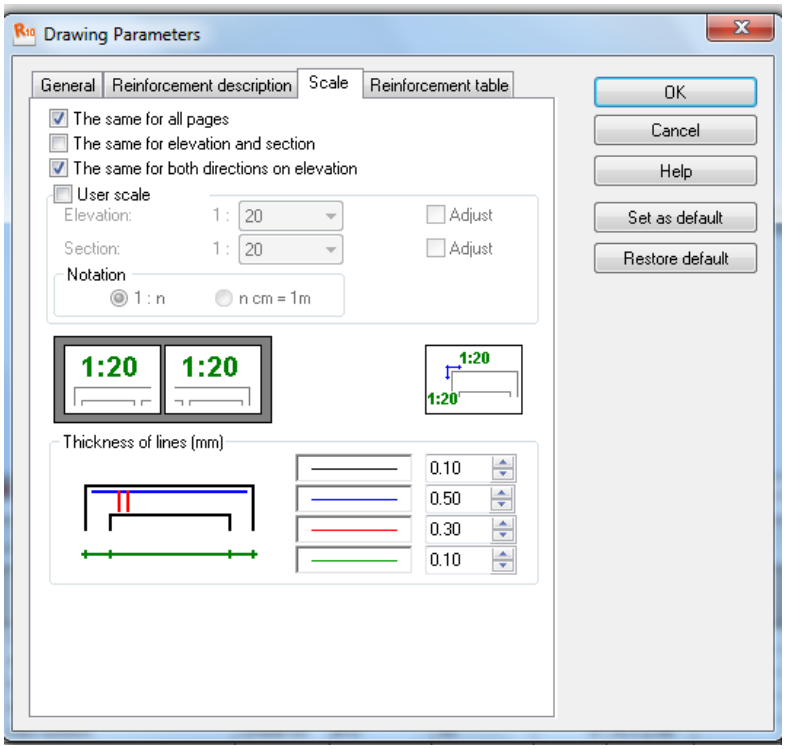




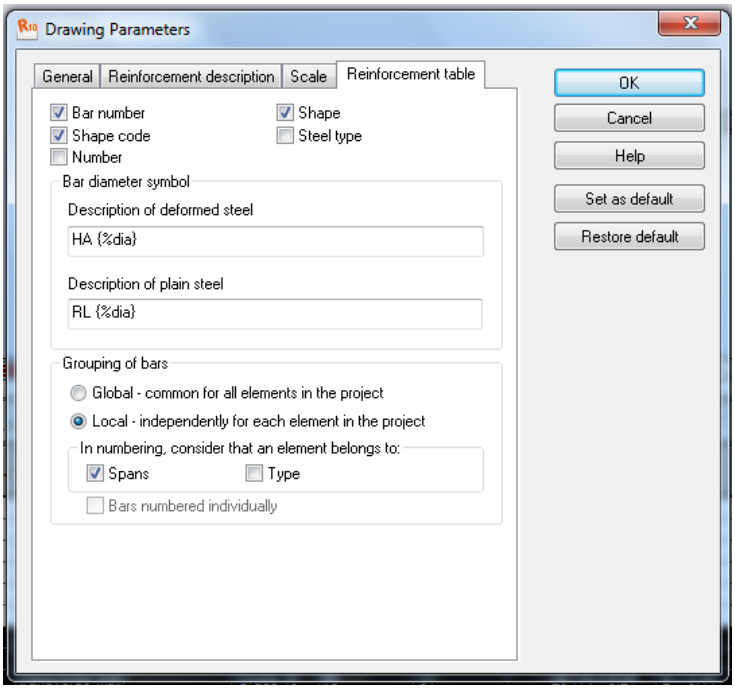
Tab Reinforcement description : ត្រង់កន្លែង Number + diameter , Length , Spacing ជ្រើសរើសយកវាគឺដើម្បីអោយវាបង្ហាញ ចំនួនដែក មុខកាត់ដែក គំលាតដែក ។

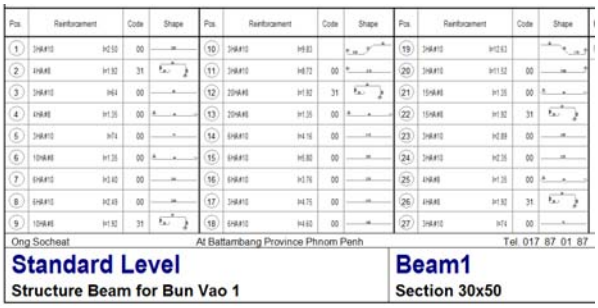


Tab Scale : សំរាប់រៀបចំពណ៌ ទំហំបន្ទាត់ រៀបចំក្រដាស់ ..... ។



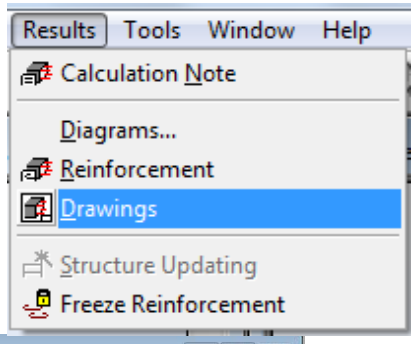
Tab Reinforcement table : សំរាប់អោយវាបង្ហាញចំនួន ដែក ទ្រង់ទ្រាយដែក ប្រភេទដែក.....។

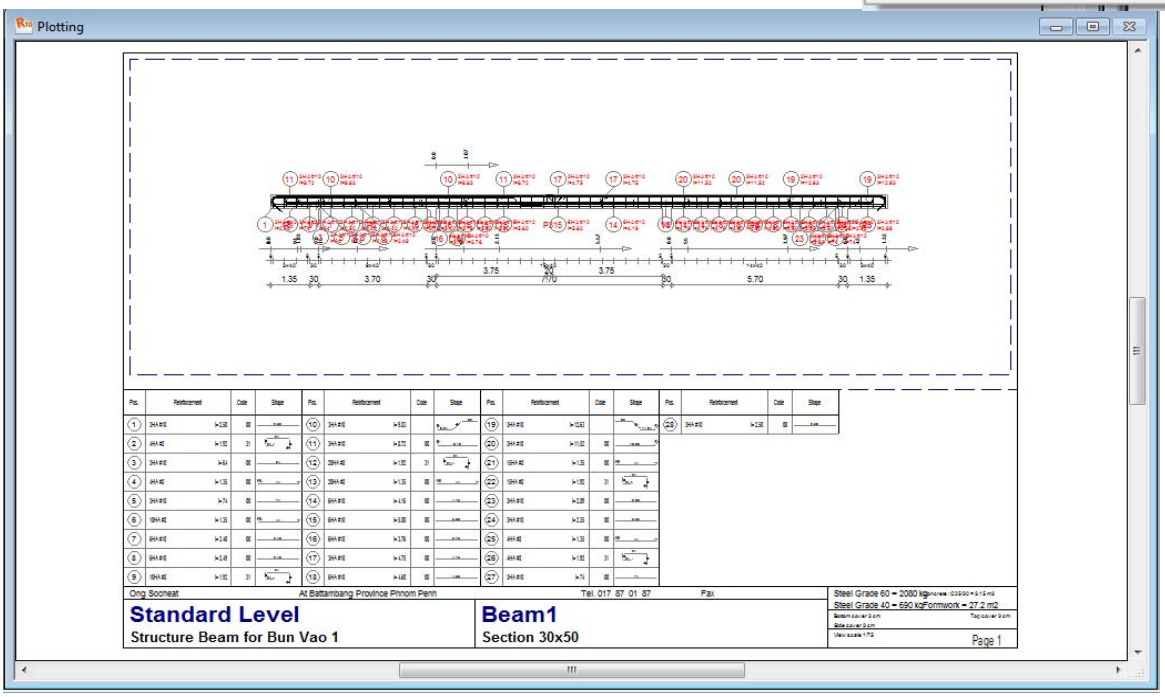




១១- មើលប្លង់ Drawings

- ដំបូងចុចលើ Icon  នេះនៅស្តាំអេក្រង់រឺ ចុចលើ Results Menu > Drawings ដូចរូប



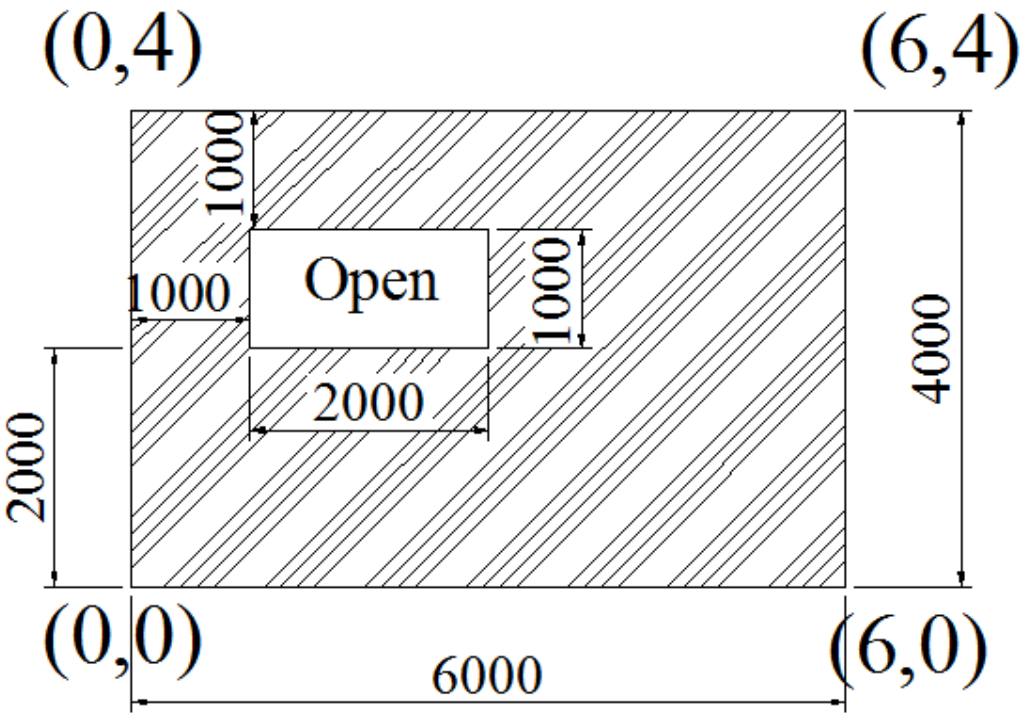


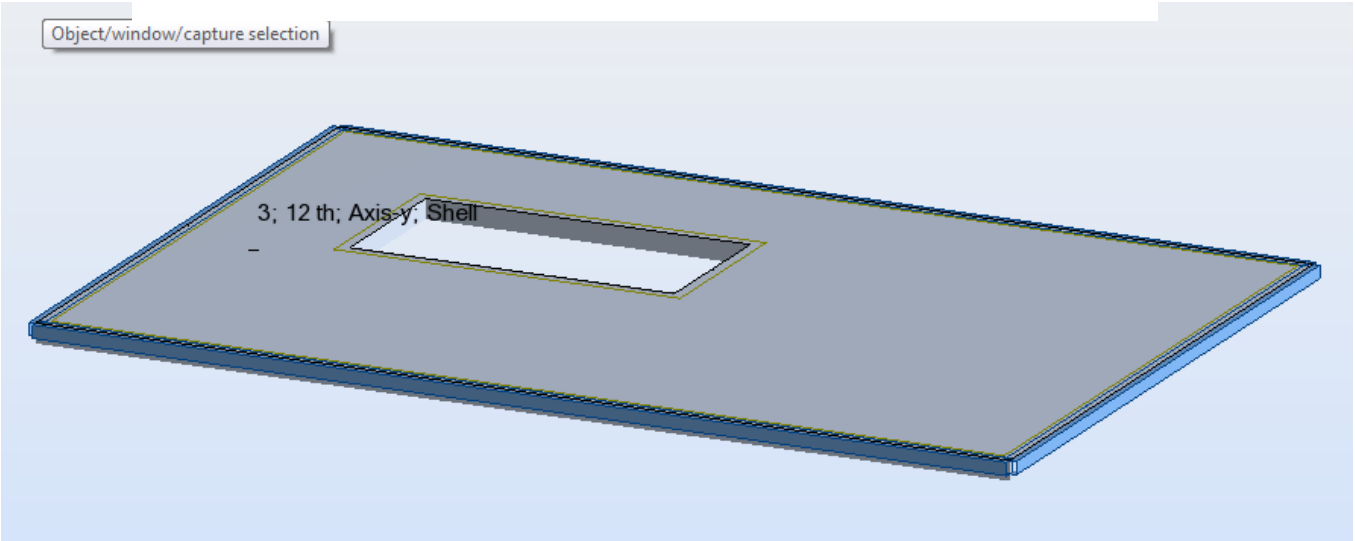
មេរៀនទី២

ការគណនាកំរាលប្តូចសេ

RC Slab Design

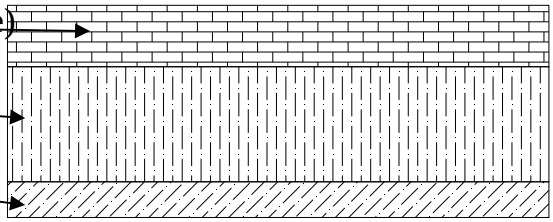
កំរាលប្តូចសេយកមកគណនា RC Slab Calculation :





3D View

- **<u>មុខកាត់គ្រឿងបន្ថំ</u>** <u>Frame Section :</u>



  - ការ៉ូ និង បាយអរ : 5cm (Mortar + Tile)
  - បង់សេកំរាស់ : 12cm (Slab)
  - បាយអរបូកពីដាន : 2cm (Plastering)

- **<u>កំណត់បន្ទុក</u>** <u>Determine loading :</u>

  - Weight of Tile f_t = 22kN/m^3
  - Weight of Mortar f_M = 22kN/m^3
  - Weight of Plastering f_p = 20kN/m^3
  - Weight of Concrete f_c = 25kN/m^3

- **<u>គណនាបន្ទុកក្នុង</u>** <u>1m^2 Calculate loading 1 m^2 :</u>

1. <u>Dead load :</u>
   - Mortar & Tile = 0.05m X 22kN/m^3 = 1.1kN/m^2
   - Plastering = 0.02m X 20kN/m^3 = 0.4kN/m^2
2. <u>Self weight :</u> (មិនចាំបាច់គណនាទេពិព្រោះ Program វាកំណត់ដោយខ្លួនវា)
   - Slab = 0.12m X 25kN/m^3 = 3kN/m^2
3. <u>Live load</u> : = 1.9kN/m^2

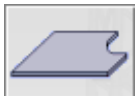
- **<u>សរុបបន្ទុក</u>** <u>Total load:</u>

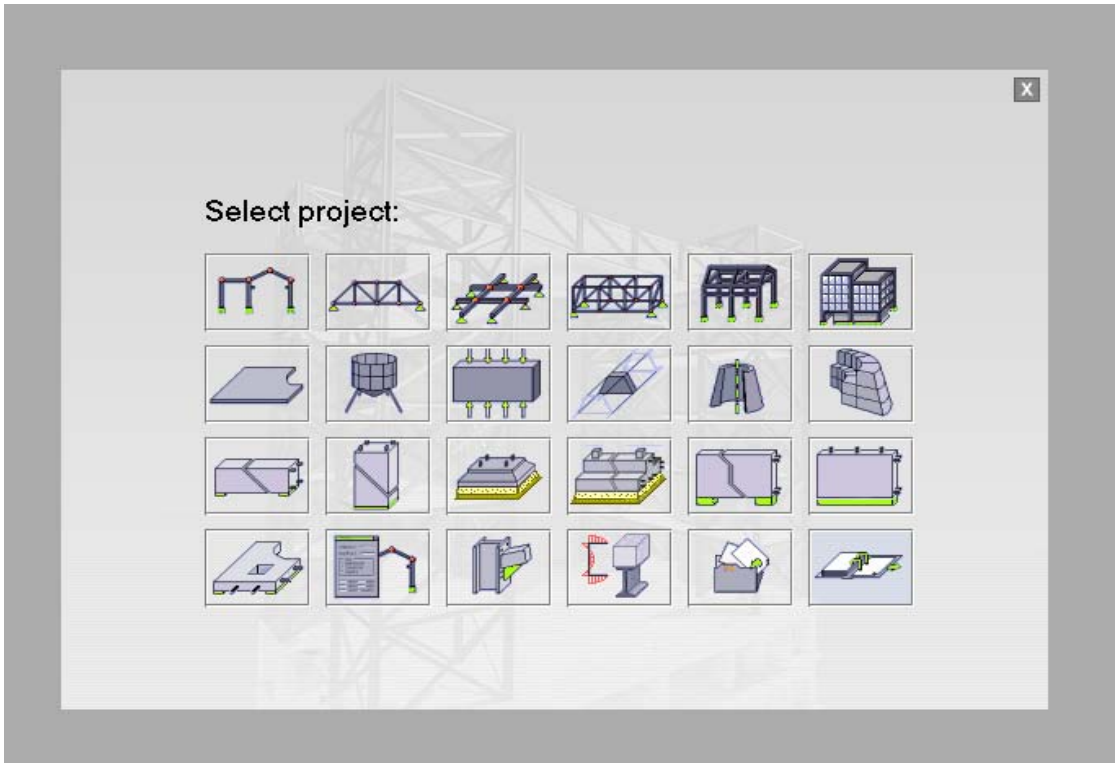
  - Dead load : =1.5kN/m^2
  - Live load : =1.9kN/m^2

- <u>Analysis and Design យើងគណនាប្រើ:</u> ACI code
- <u>Material Properties for Analysis and Design យើងប្រើ Euro code :</u>

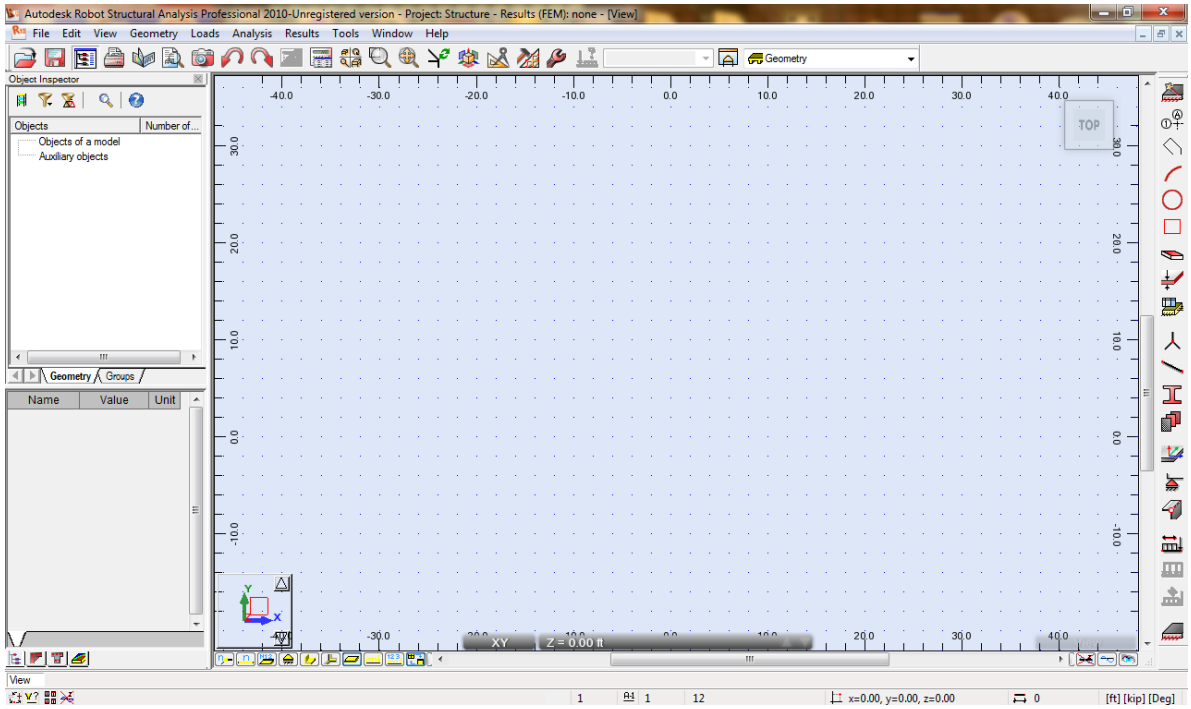
- Yield Strength of Longitudinal reinforcement : 400 MPa
- Yield Strength of Transversal reinforcement : 235 MPa
- Compressive Strength of Concrete (Cylinder) : 25 MPa
- Cover : 2cm

Step by Step Solution

នៅពេលបើក Program Autodesk Robot Structural Analysis Professional 2010 ដំបូងយើងនិងឃើញដូចរូបខាងក្រោមបន្ទាប់មកយើងជ្រើសរើសយក Icon  ដូចដែលបានបង្ហាញនៅលើរូបខាងក្រោម :

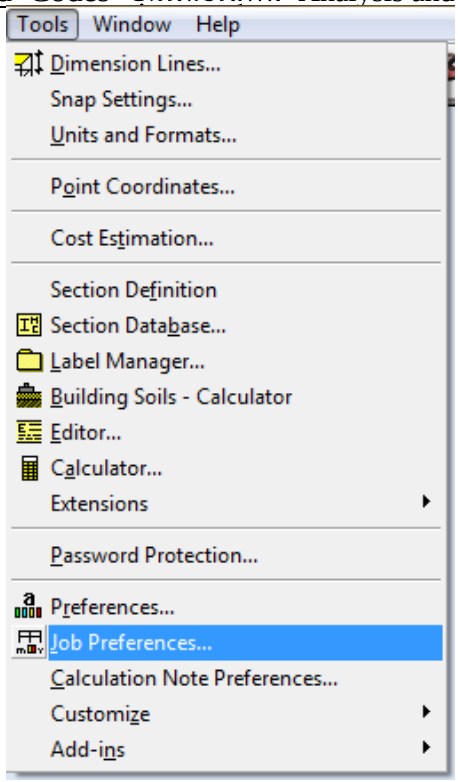


បន្ទាប់មកទៀតអេក្រង់នឹងបង្ហាញដូចរូបខាងក្រោមនេះ

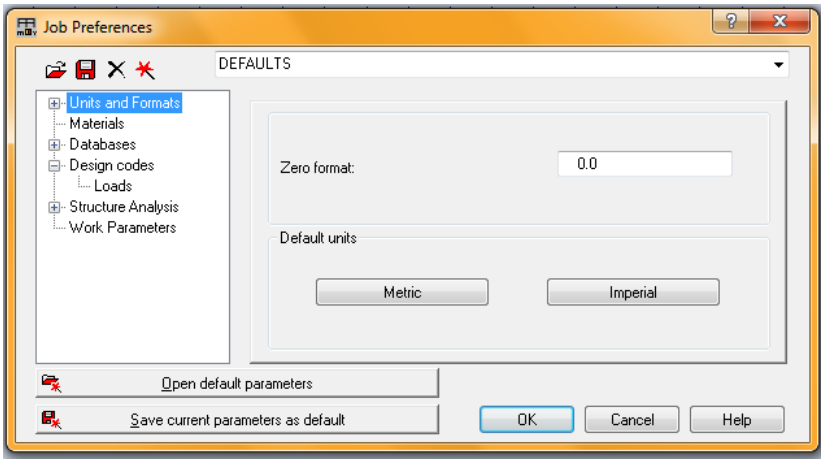


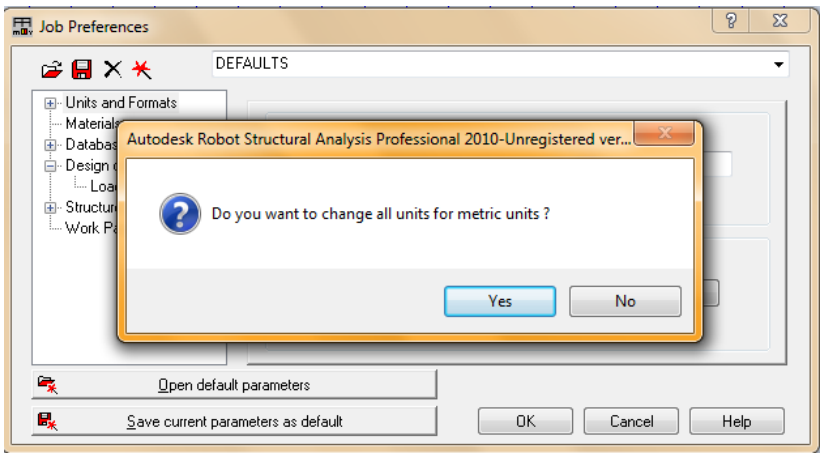
- **បន្ទាប់មកចូរកំណត់** Units and Format, Materials and Codes មុនពេលធ្វើការ Analysis and Design :

ចុចលើ Tool Menu > Job Preference……



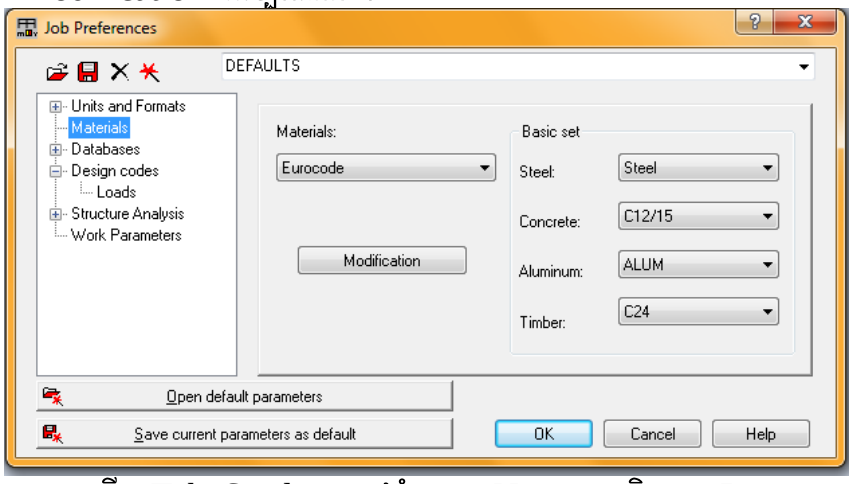
1. អេក្រង់នឹងបង្ហាញផ្ទាំងដូចរូប នៅត្រង់ Units and Formats ត្រូវកំណត់វាជា Metric រួច Yes



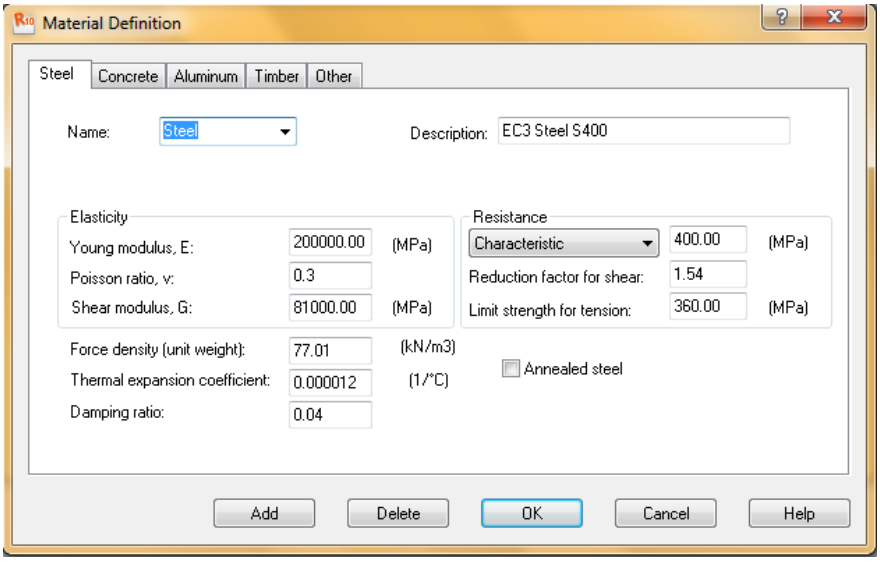


www.yerngshare.blogspot.com

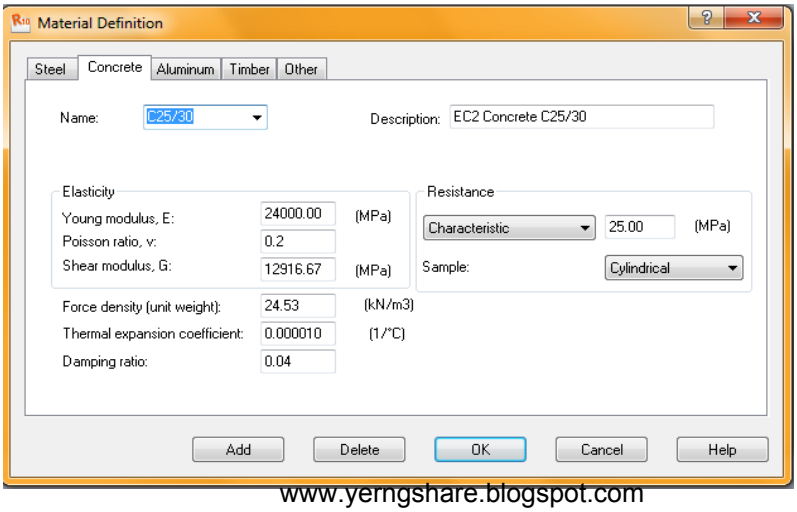
2. នៅត្រង់ Materials ត្រូវកំណត់កូដអោយ Materials ត្រូវយកកូដ Euro code រួចចុចទៅលើ Modification ដើម្បីកំណត់ ។



3. រួចចុចលើ Tab Steel រួចត្រូវបំពេញ Name និង Description ដូចខាងក្រោម

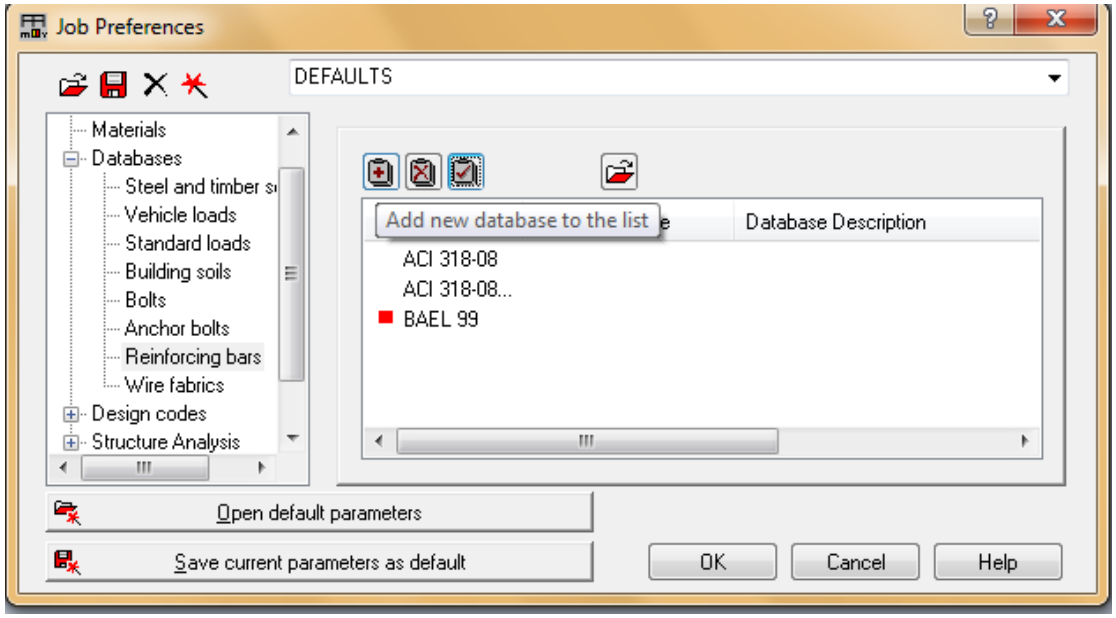


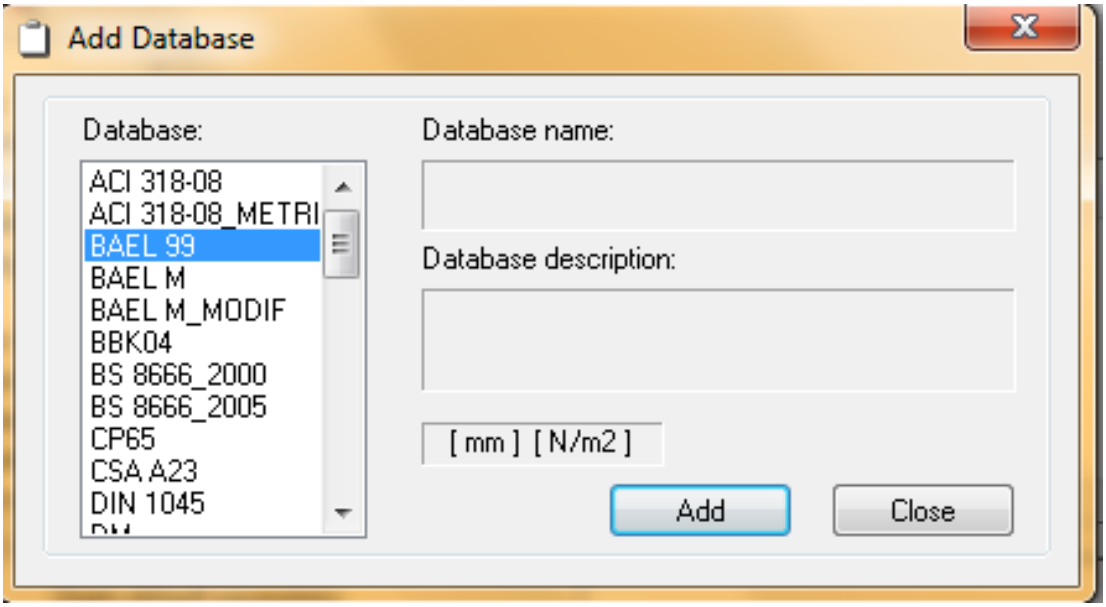
4. រួចចុចលើ Tab Concrete រួចត្រូវបំពេញ Name និង Description និង Young modulus ដូចខាងក្រោម រួចចុច OK ។



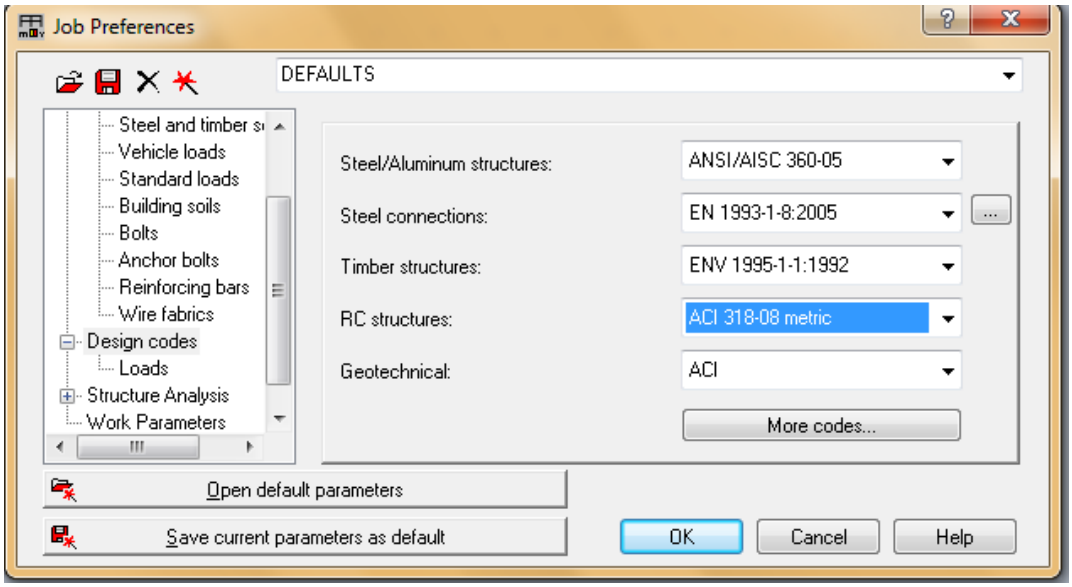
www.yerngshare.blogspot.com

5. រួចចុចលើ Databases រួចរើសយក Reinforcing bars ហើយនៅប្រអប់ខាងស្តាំ ត្រូវបង្កើតកូដ BAEL99 ដោយចុចទៅលើ Icon ដែលមានឈ្មោះថា Add new databases to the list នោះវានឹងចេញតារាង Add Databases រួចរើសយក BAEL99 រួចចុច add ។ បន្ទាប់មករើសកូដដោយយក Mouse select on code រួចចុច Icon ដែលមានឈ្មោះថា Set as the current database ។

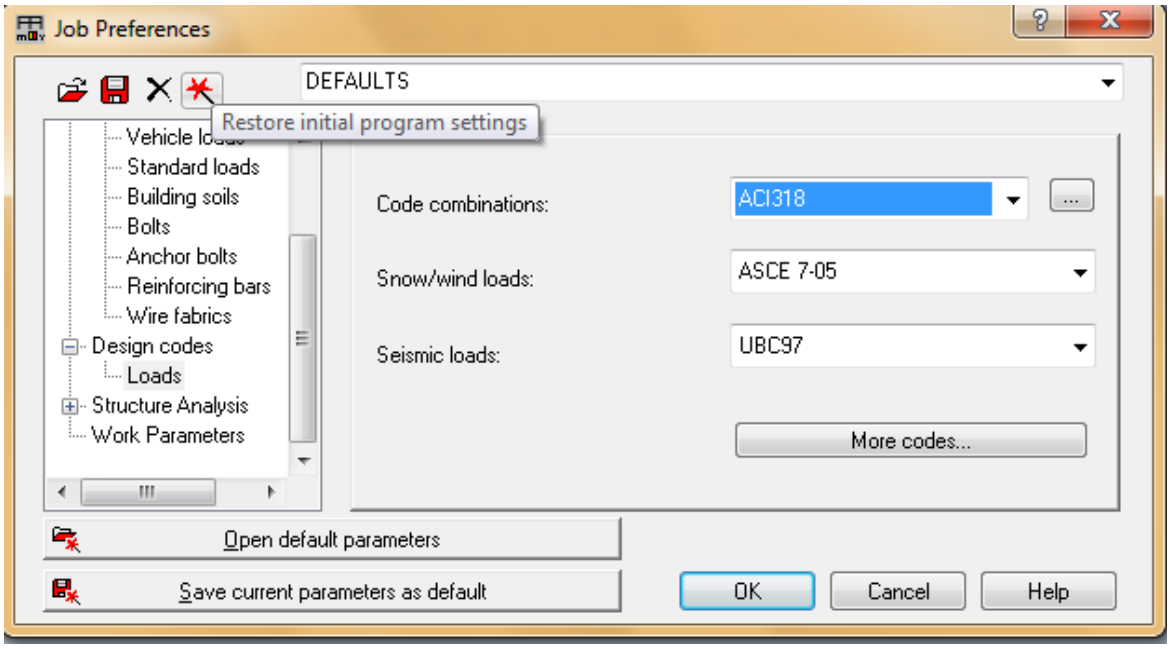




6. រួចចុចលើ Design codes ក្នុងប្រអប់ RC Structures ត្រូវកំណត់ជា ACI 318-08 metric



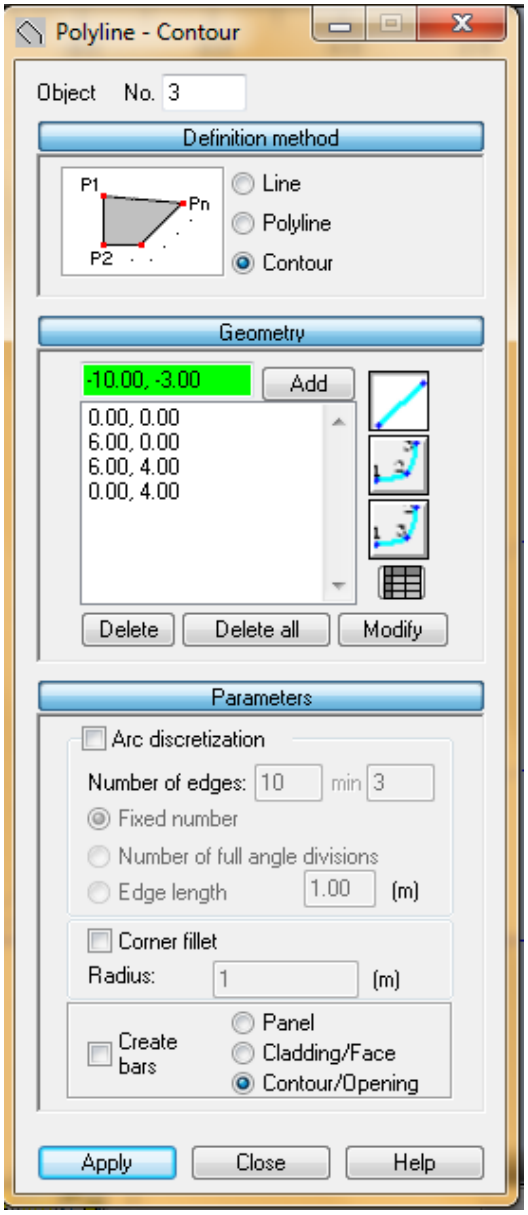
7. នៅត្រង់ Loads ក្នុងប្រអប់ Code combinations ត្រូវកំណត់ជា ACI318 រួចចុច OK ។

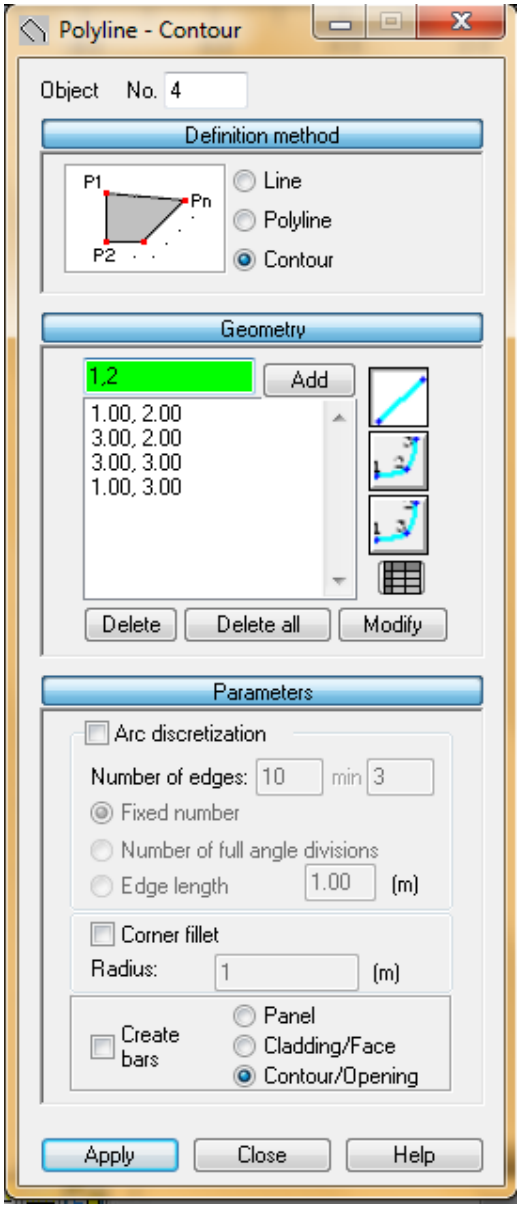


I. ការកំណត់ការគួរកំរាលប្លង់សេ

ដំបូងត្រូវចូល Polyline -Contour ហើយនៅត្រង់ Definition method ត្រូវយក Contour រួចនៅត្រង់ Geometry ត្រូវបំពេញដូចខាងក្រោម : បន្ទាប់មកត្រូវចុច Apply រួចចុច Close ។

កំរាលប្លង់សេ (x, y) ត្រូវចុច	កន្លែងជណ្តើរ (x, y) ត្រូវចុច
(0,0) រួចចុច add	(1,2) រួចចុច add
(6,0) រួចចុច add	(3,2) រួចចុច add
(6,4) រួចចុច add	(3,3) រួចចុច add
(0,4) រួចចុច add	(1,3) រួចចុច add
(0,0) រួចចុច add	(1,2) រួចចុច add

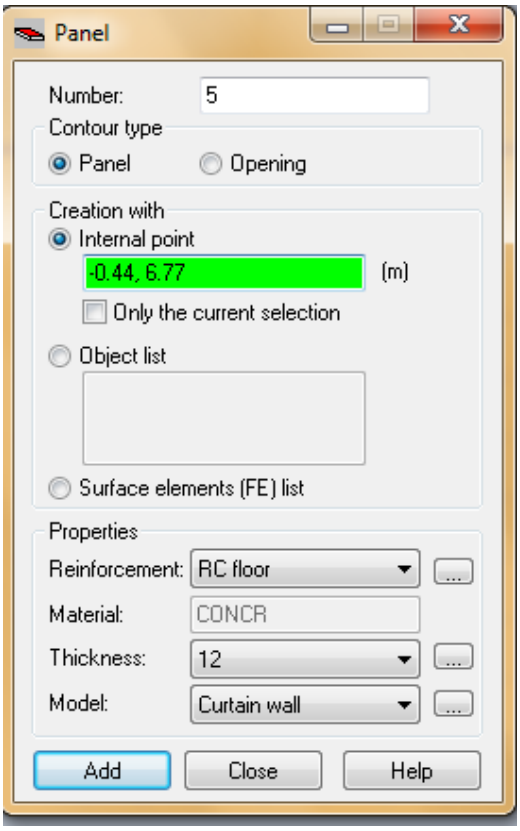


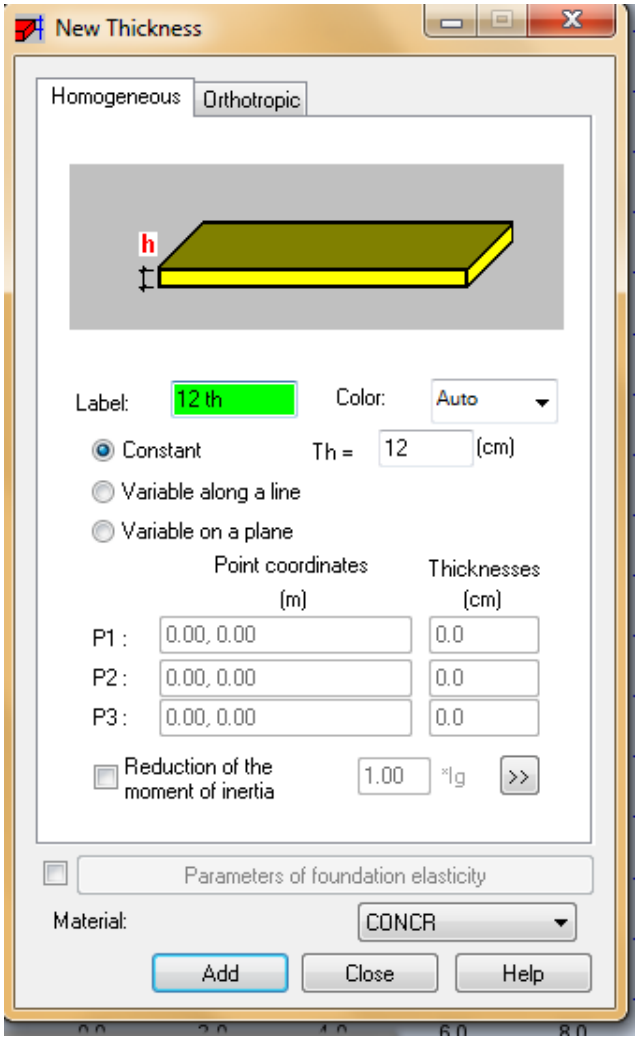


II. ការកំណត់កំរាស់កំរាលថ្ពង់សេ Slab Thickness

ដំបូងត្រូវចូល Panel រួចចូល New Thickness

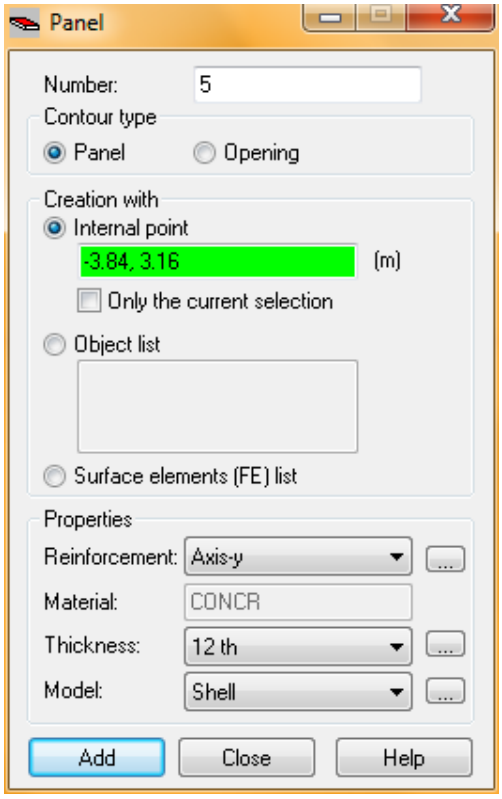
Slab Thickness = 12cm រួចចុច add and close ។

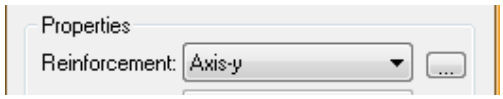






បន្ទាប់ពីកំណត់កំរាស់ប្លង់សេរួចហើយ យើងចូលទៅចុច Reinforcement ដើម្បីរៀបចំដែកនឹងកូដ ។



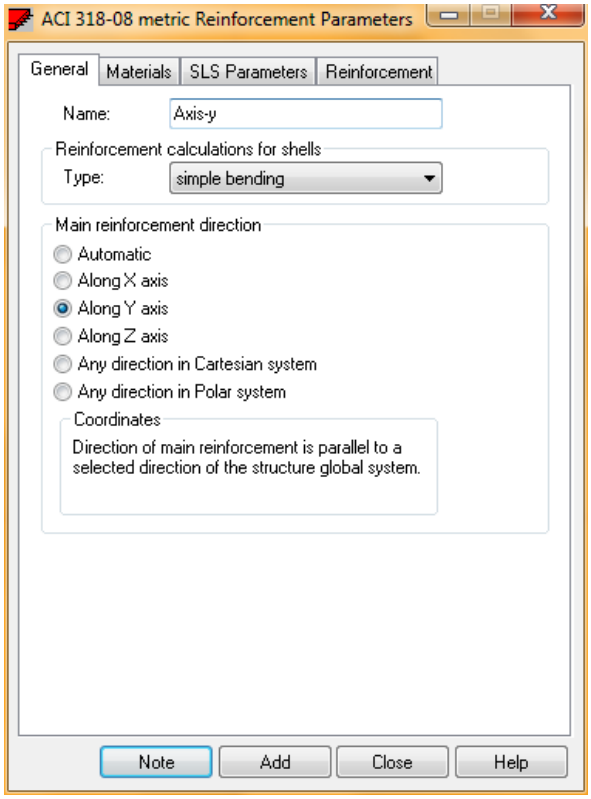


នៅក្នុង Reinforcement មានដូចខាងក្រោម

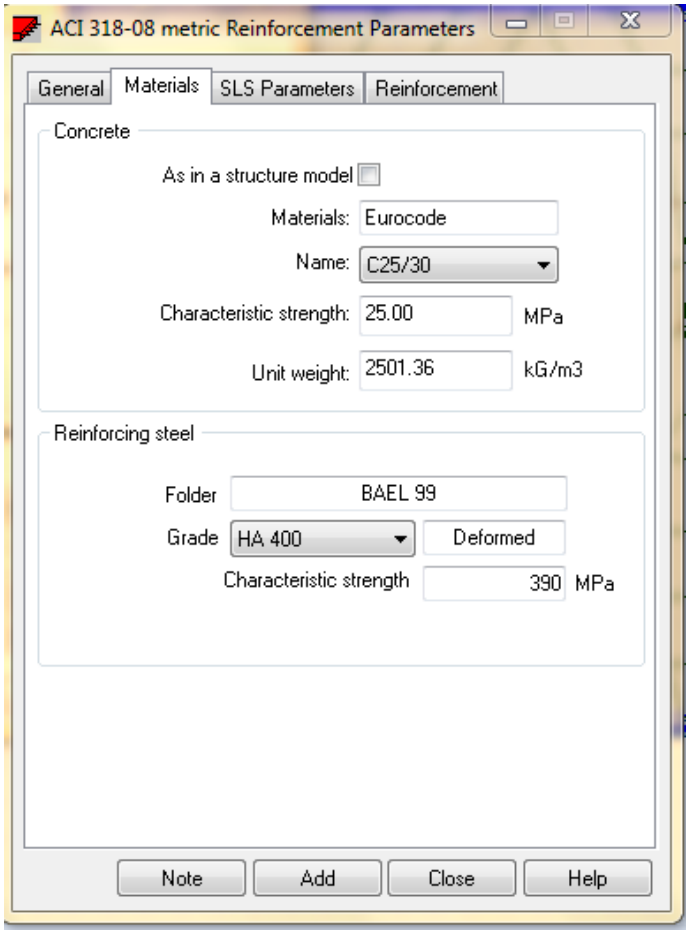
- General
- Materials
- SLS Parameters
- Reinforcement

■ នោះវានឹងបង្ហាញតារាងដូចខាងក្រោម :

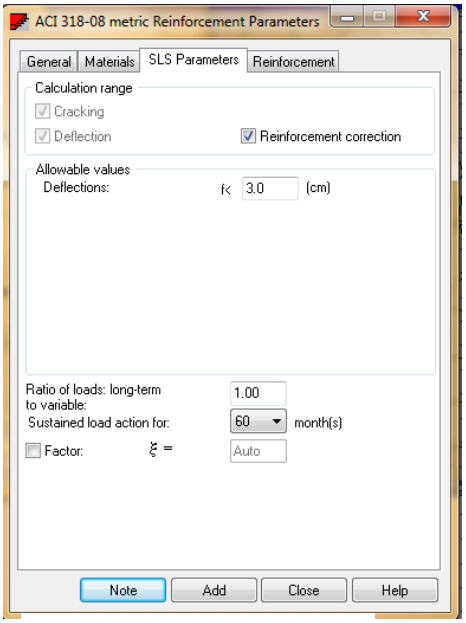
Tab General



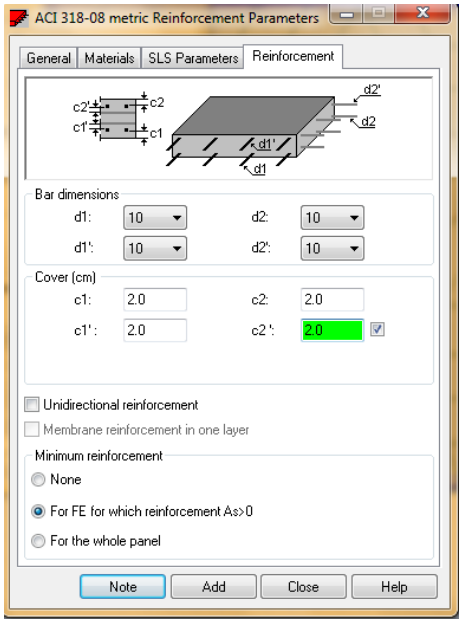
Tab Materials



Tab SLS Parameters

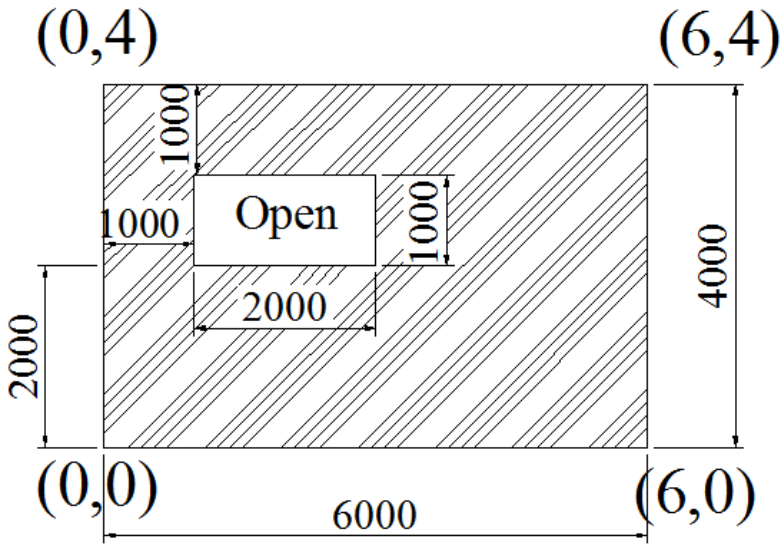


Tab Reinforcement



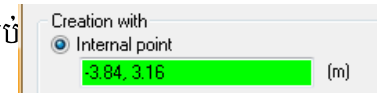


\+ បង្កើត Coordinate

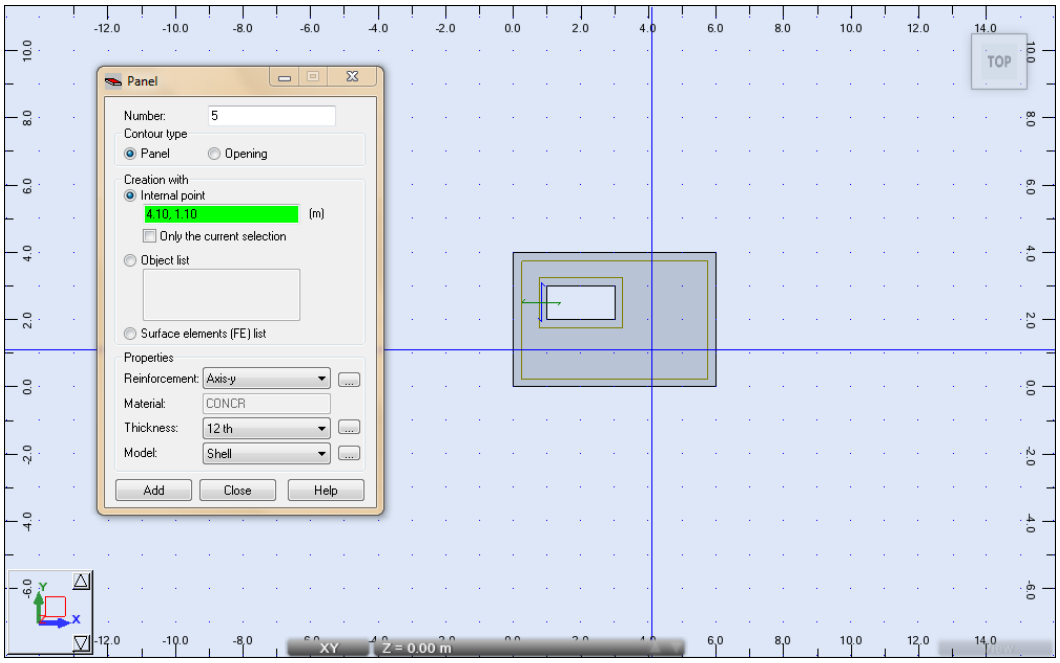


\+ Poly line

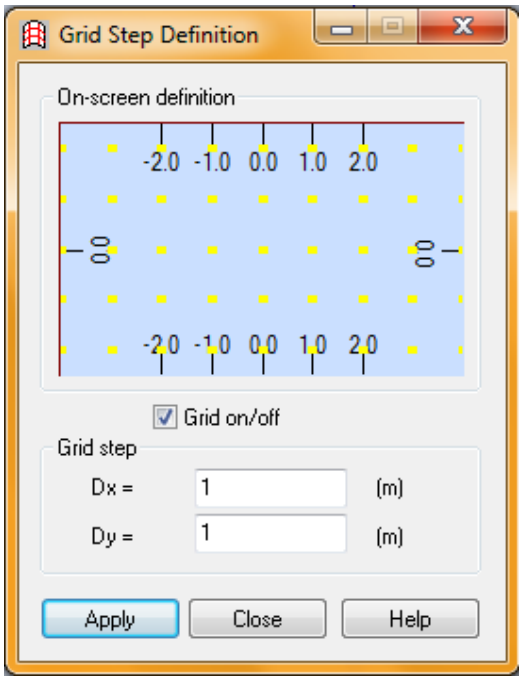
កំរាលប្លង់សេ (x, y) ត្រូវចុច		កន្លែងជណ្តើរ (x, y) ត្រូវចុច	
(0,0) រួចចុច Insert		(1,2) រួចចុច Insert	
(6,0) រួចចុច Insert		(3,2) រួចចុច Insert	
(6,4) រួចចុច Insert	រួចចុច Add	(3,3) រួចចុច Insert	រួចចុច Add
(0,4) រួចចុច Insert		(1,3) រួចចុច Insert	
(0,0) រួចចុច Insert		(1,2) រួចចុច Insert	

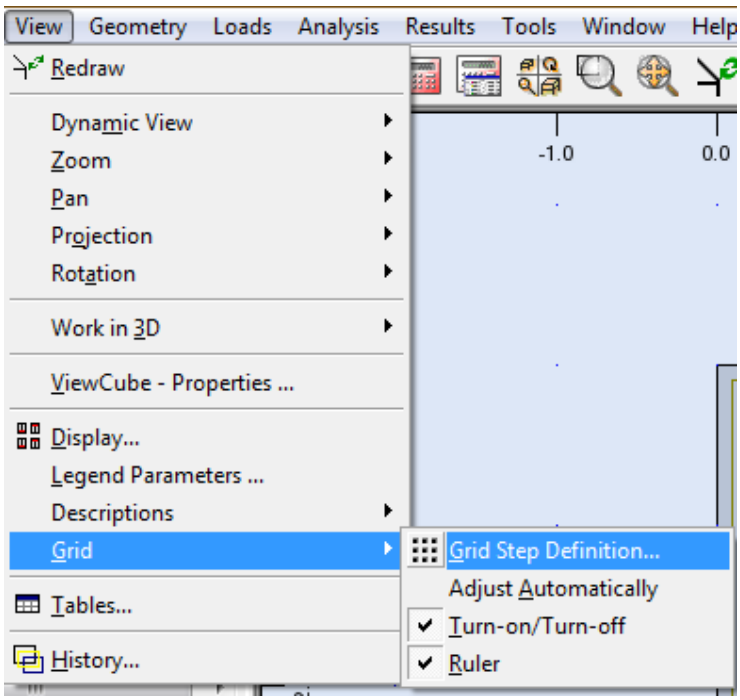
- បន្ទាប់ពីបំពេញរួចរាល់ហើយ ត្រូវយក Mouse មកដាក់នៅក្នុងប្រអប់ 

រួចត្រូវចុចនៅក្នុងប្លង់សេ (ដូចរូប) នោះប្លង់សេនឹងចាក់ហើយជាការស្រេច ។



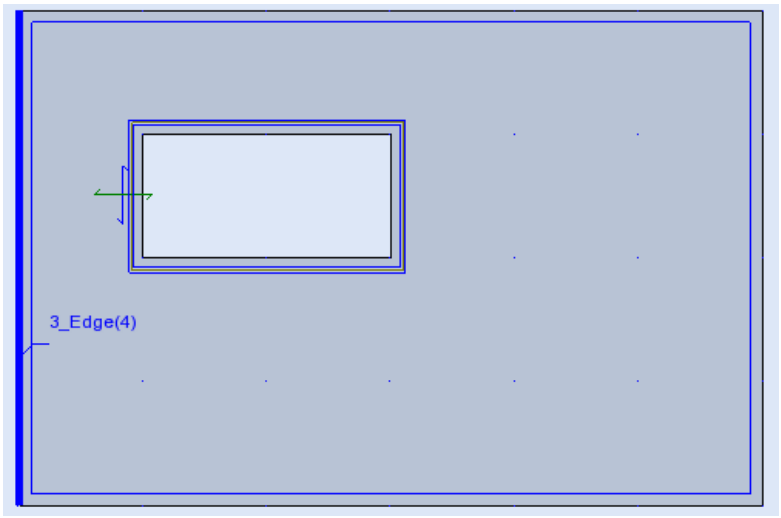
- ការកំណត់ Grid Step Definition សំរាប់ Mesh ។ ដំបូងត្រូវចុច View Menu រួចចុចលើ Grid រួចចុចលើ Grid Step Definition ។(ដូចរូប)

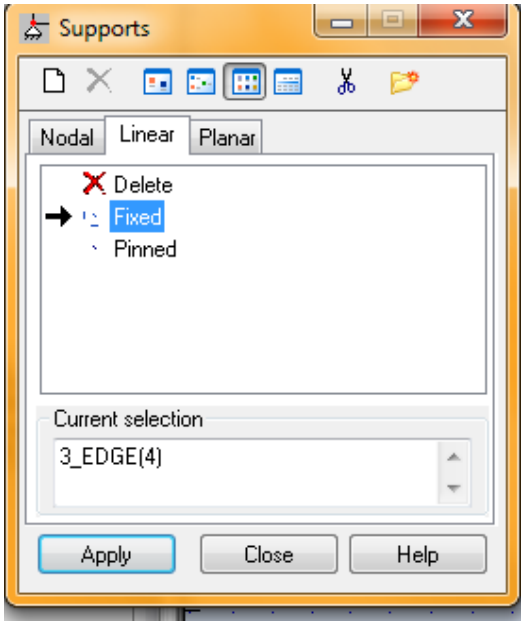




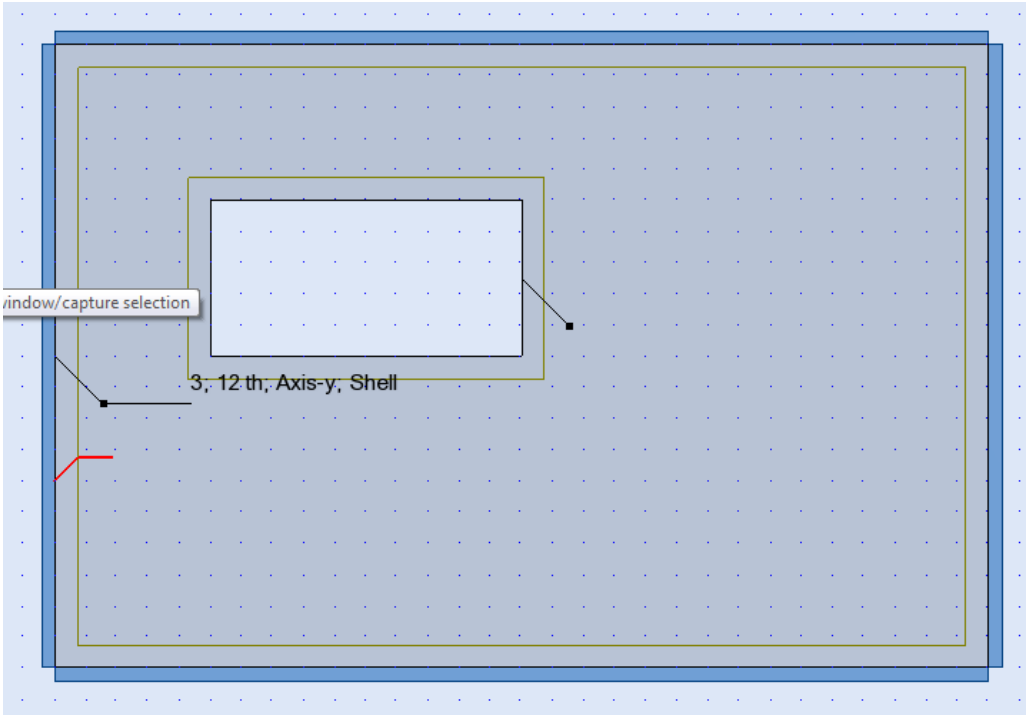
IV ការកំណត់ទំរ និងការដាក់បន្ទុក Determine Supported :

- ដំបូងត្រូវចុចលើ Icon  នៅស្តាំអេក្រង់ ។ ត្រូវកំណត់បន្ទុកជា Fixed ដាក់ជុំវិញកំរាលបួងសេ ត្រូវកំណត់វាជា Linear គឺត្រូវដាក់ Mouse នៅក្នុងប្រអប់ Current Selection រួចយកទៅ Select លើគែមបួងសេទាំងបួនទិស ទាល់តែវាចេញ ពណ៌ខៀវចាំ Select វាជាក្រោយ ។



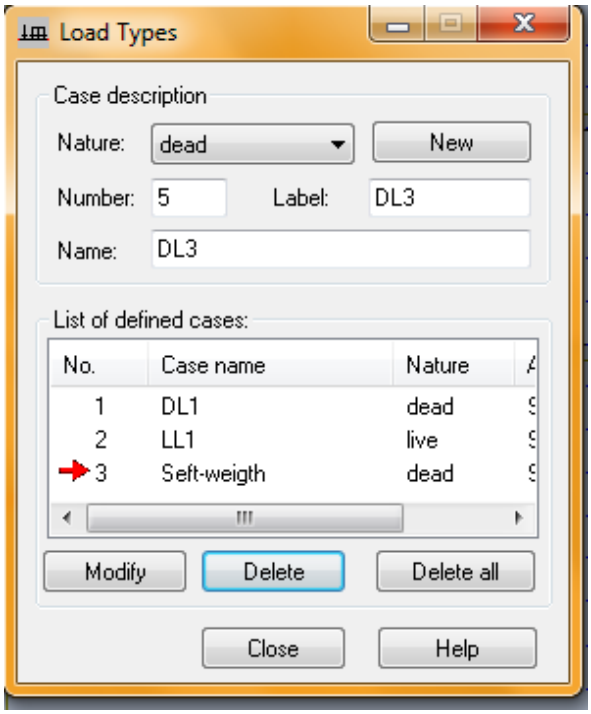


- បន្ទាប់ពីដាក់បន្ទុករួច វានឹងចេញរូបដូចខាងក្រោម : ជុំវិញវាមានបន្ទាត់ពន្លឺជុំវិញវា ។(ដូចរូប)



V បង្កើតប្រភេទបន្ទុក (Load Types) ៖

- ដំបូងត្រូវជ្រើសរើសបន្ទុកនៅកន្លែង Nature រួចដាក់ឈ្មោះនៅកន្លែង Name រួចចុច New ។ បញ្ជាក់បើ Nature រើសជាអ្វីនោះ នៅត្រង់ Name ត្រូវដាក់ឈ្មោះដូចនោះដែល ។ (ដូចរូប)



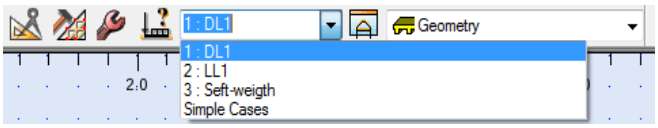
Nature : dead
Name : DL1 New

Nature : live
Name : LL1 New

Nature : dead
Name : Self- weight New

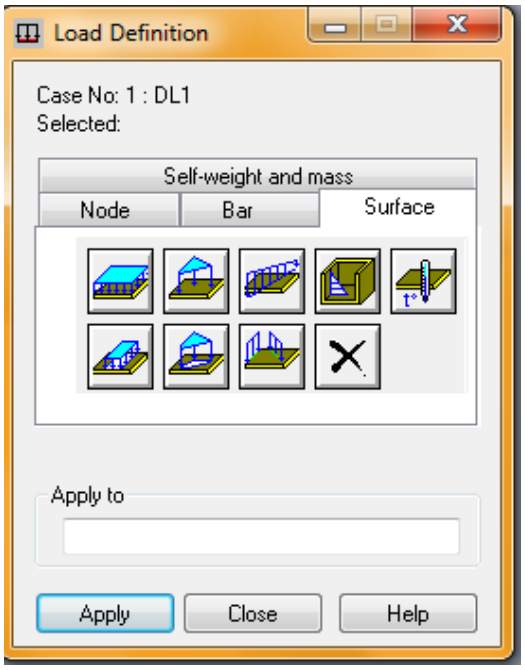
VI កំណត់ការដាក់បន្ទុកលើកំរាលច្រង់សេ (Apply Loads) ៖

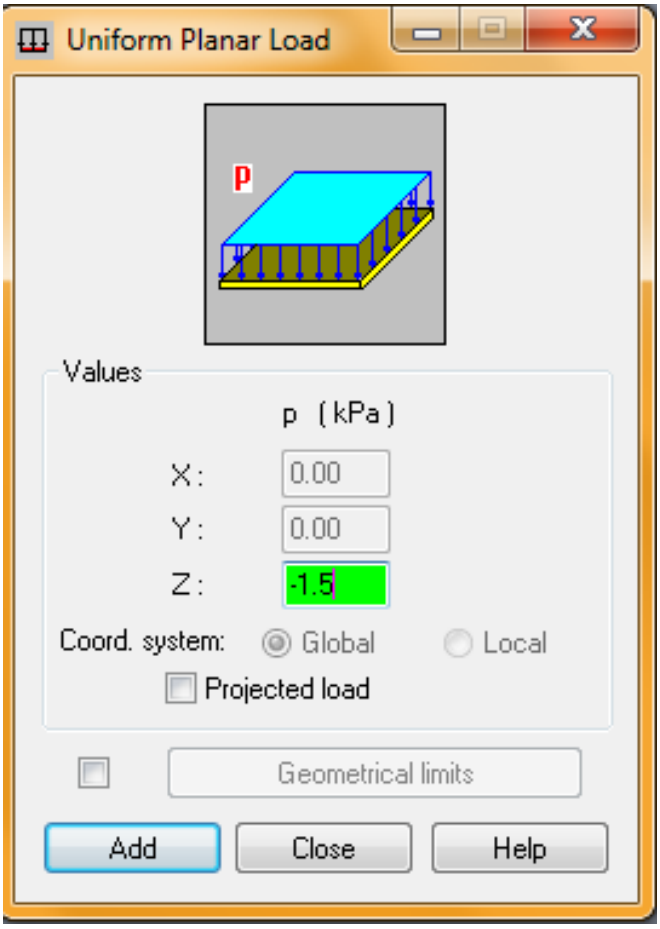
~ ដំបូងត្រូវចុចលើ Icon  នៅខាងស្តាំអេក្រង់ ។ បើដាក់ Dead Load គឺត្រូវរើស DL1 នៅក្នុងប្រអប់ខាងលើដូចរូប



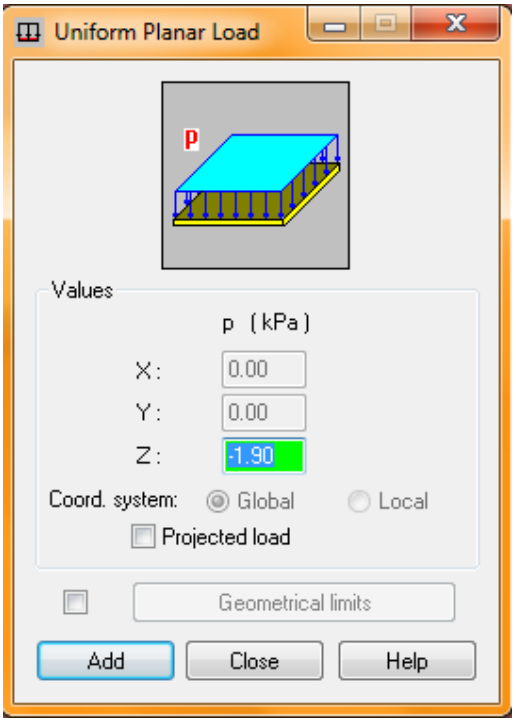
- បន្ទាប់មកត្រូវចុចលើ Surface រួចចុចលើ Icon  ដូចរូប ។ បន្ទាប់មកត្រូវបញ្ចូល Dead Load គឺ Z = -1.5 វាយសញ្ញាដកគឺបញ្ជាក់ថាទិសដៅចុះក្រោម ។ រួចចុច add រួចដាក់ Mouse នៅក្នុងប្រអប់ Apply to រួចយក មកចុះលើបង្ហង់សេ រួចចុច Apply and close នោះជាការស្រេច ។ បន្ទាប់មកដាក់ Live Load ក៏ដូចនោះដែល ។

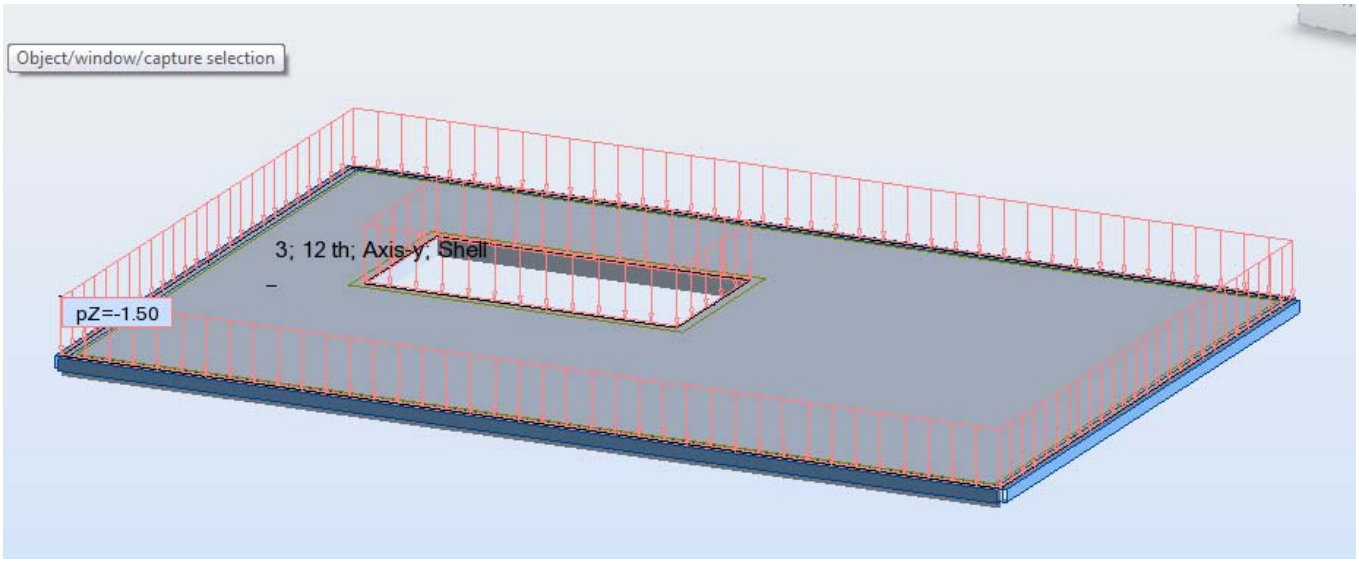
- ការដាក់បន្ទុក Dead Loads គឺ Z = -1.5kN/m^2



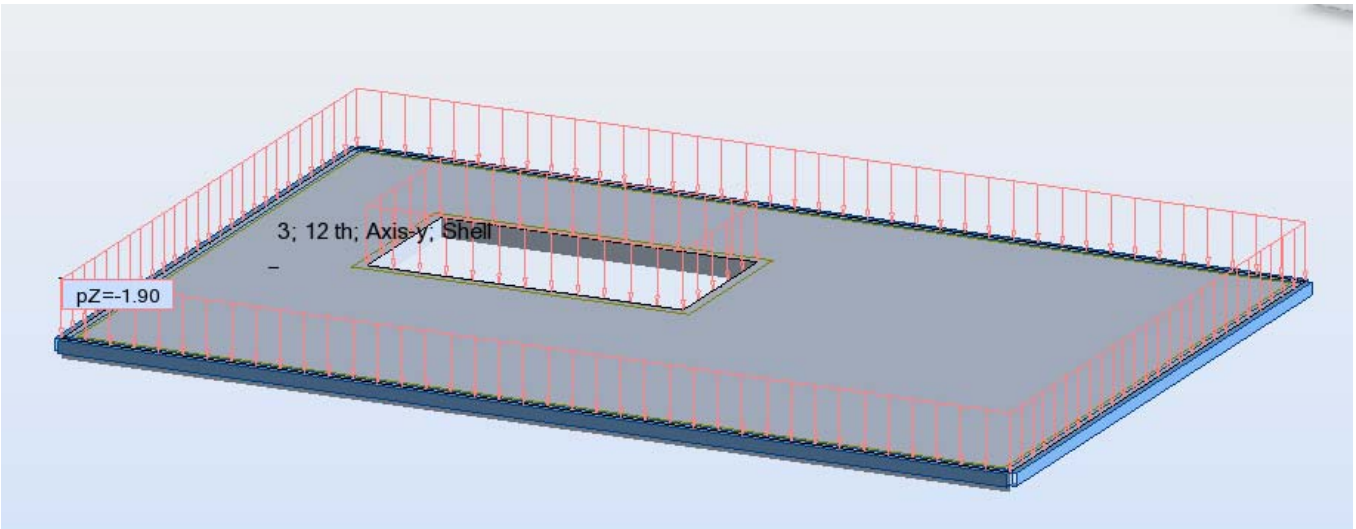


- ការដាក់បន្ទុក Live Load គឺ Z = -1.9kN/m^2





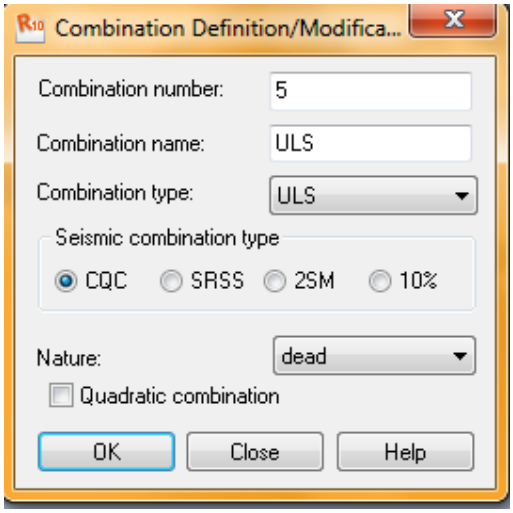
បន្ទុកអចេរ DL

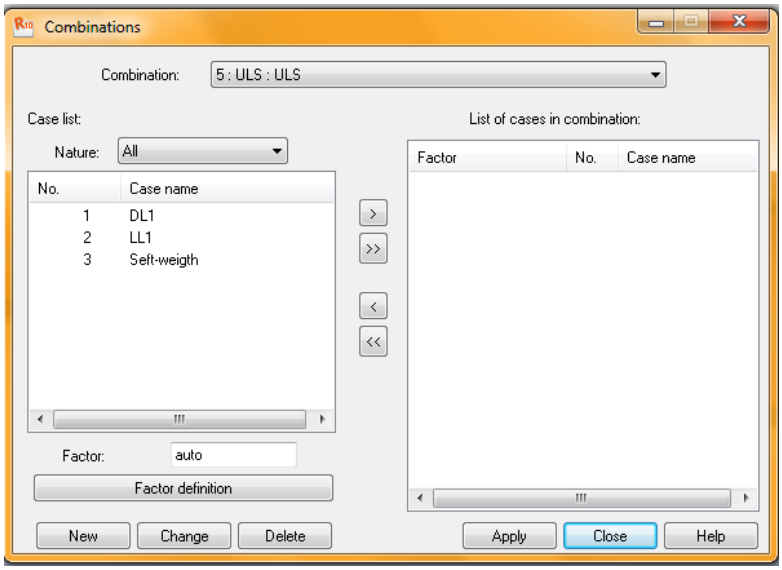


បន្ទុកថេរ LL

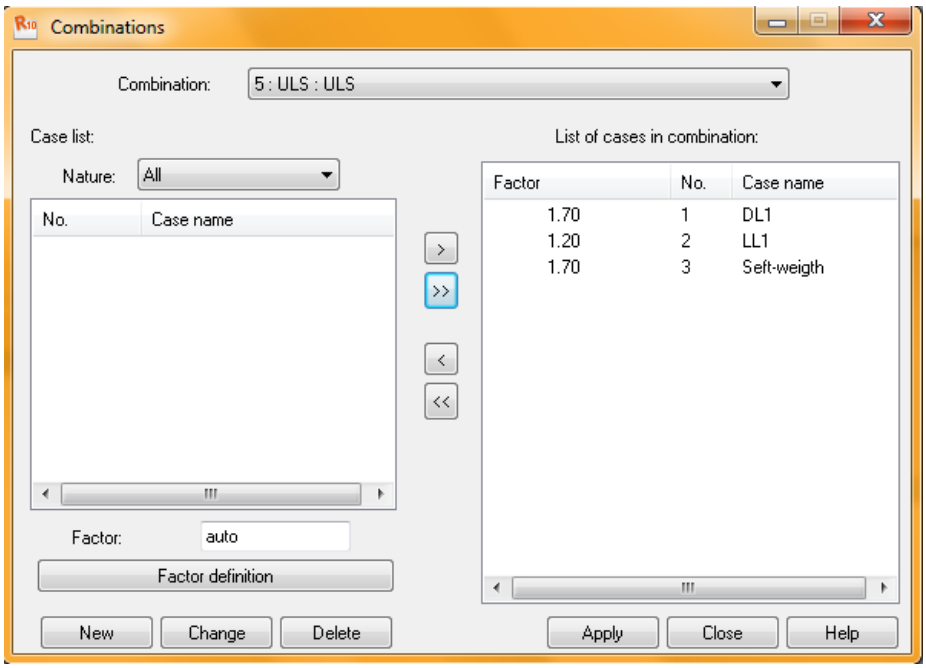
VII **ការបន្សុំបន្ទុក** (Combination Loads) ៖

- ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំអេក្រង់ ។ ត្រង់ប្រអប់ Combination Name: ត្រូវវាយ ULS ដើម្បីមើលម៉ូម៉ង់សរុបទាំងមូល ។ រួចចុច ok នោះវានឹងចេញផ្ទាំង Combination ដូចខាងក្រោម ៖

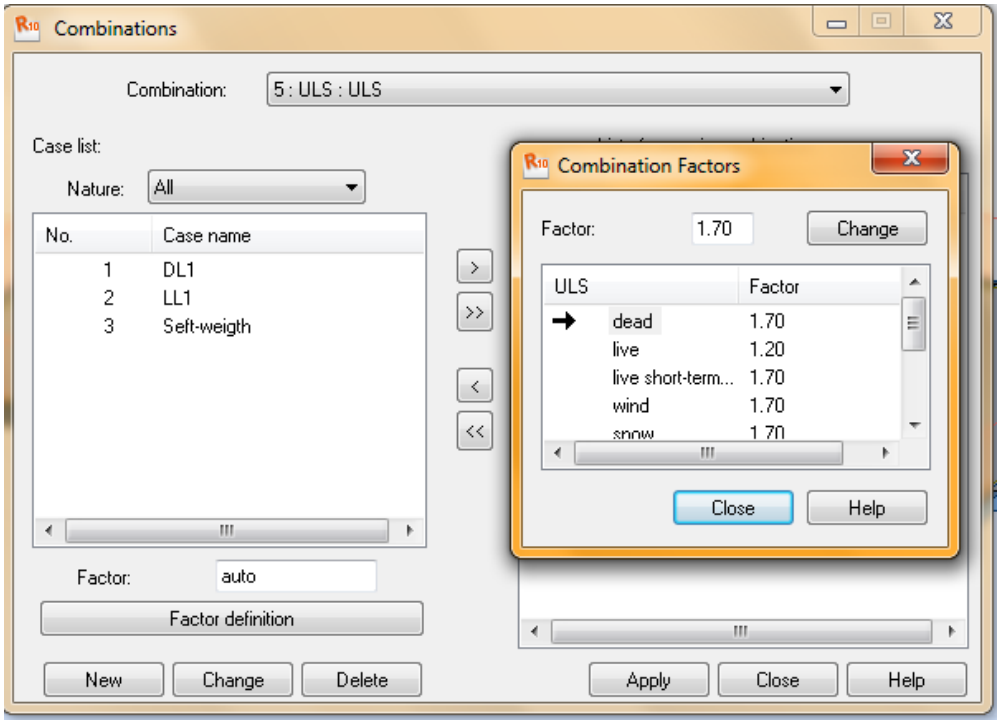




- រួចត្រូវបោះតំលៃនៅប្រអប់ខាងឆ្វេងទៅខាងស្តាំទាំងអស់ដោយចុចសញ្ញានេះ  បន្ទាប់មកវានឹងចេញផ្ទាំងដូចខាងក្រោម

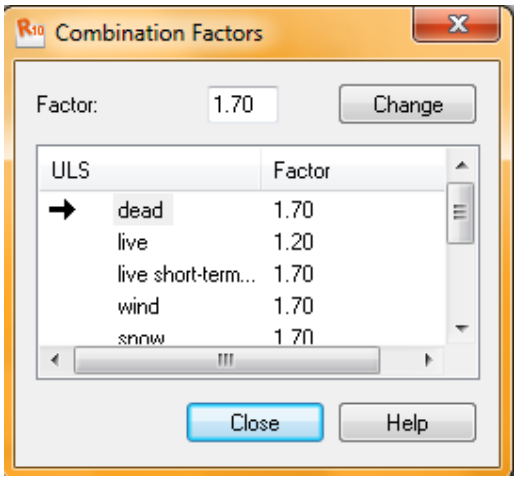


- តាមតារាងខាងលើនេះ យើងឃើញថាតំលៃបន្ទុកទាំងអស់គឺខុសទាំងអស់ ដូចនេះយើងត្រូវធ្វើការផ្លាស់ប្តូរដោយចុចសញ្ញានេះ << ឱ្យវាមកចូលក្នុងតារាងខាងឆ្វេងវិញ ដូចរូប រួចចុចពាក្យ Factor definition នោះវានឹងចេញតារាងមួយទៀតគឺ Combination Factors ដូចរូប ។

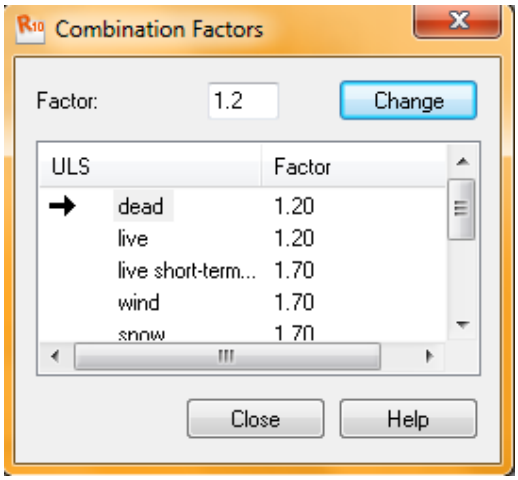


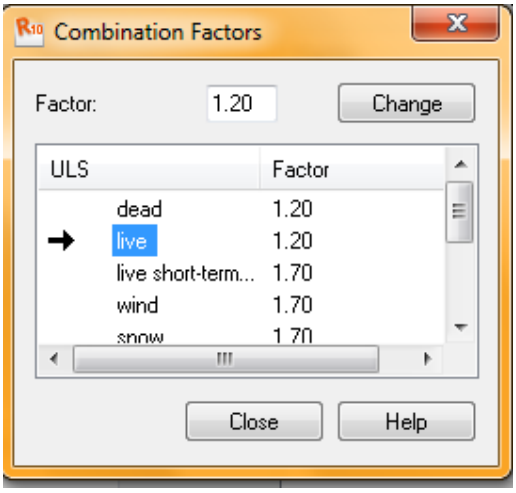
~ បន្ទាប់មកត្រង់កន្លែង Factor: ត្រូវវាយតំលៃលេខចូល ទៅតាមបន្ទុក ហើយត្រូវឱ្យសញ្ញាព្រួញដាក់ឱ្យចំបន្ទុកដែលយើងចង់ដាក់ រួចចុច Change ជាការស្រេច ។ បន្ទុកផ្សេងៗទៀតធ្វើការកែដូចគ្នា ។ មើលរូប

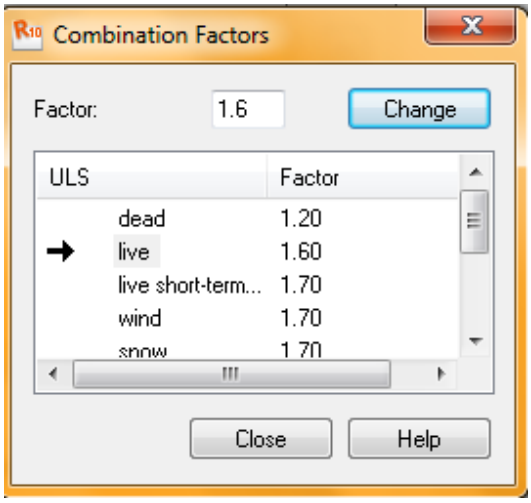
មិនទាន់ផ្លាស់ប្តូរ



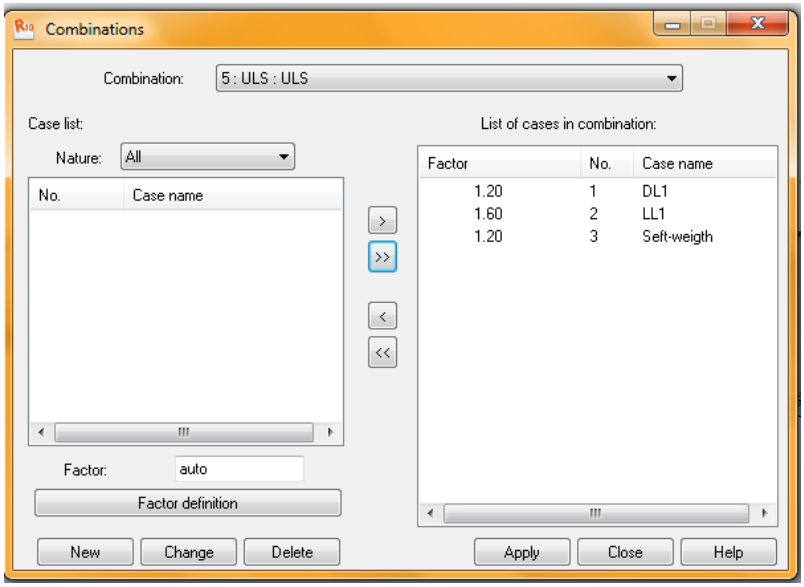
ផ្លាស់ប្តូររួច







តារាងដែលកែរួច
ហើយបោះទៅខាងស្តាំ
រួចចុច Apply and Close.

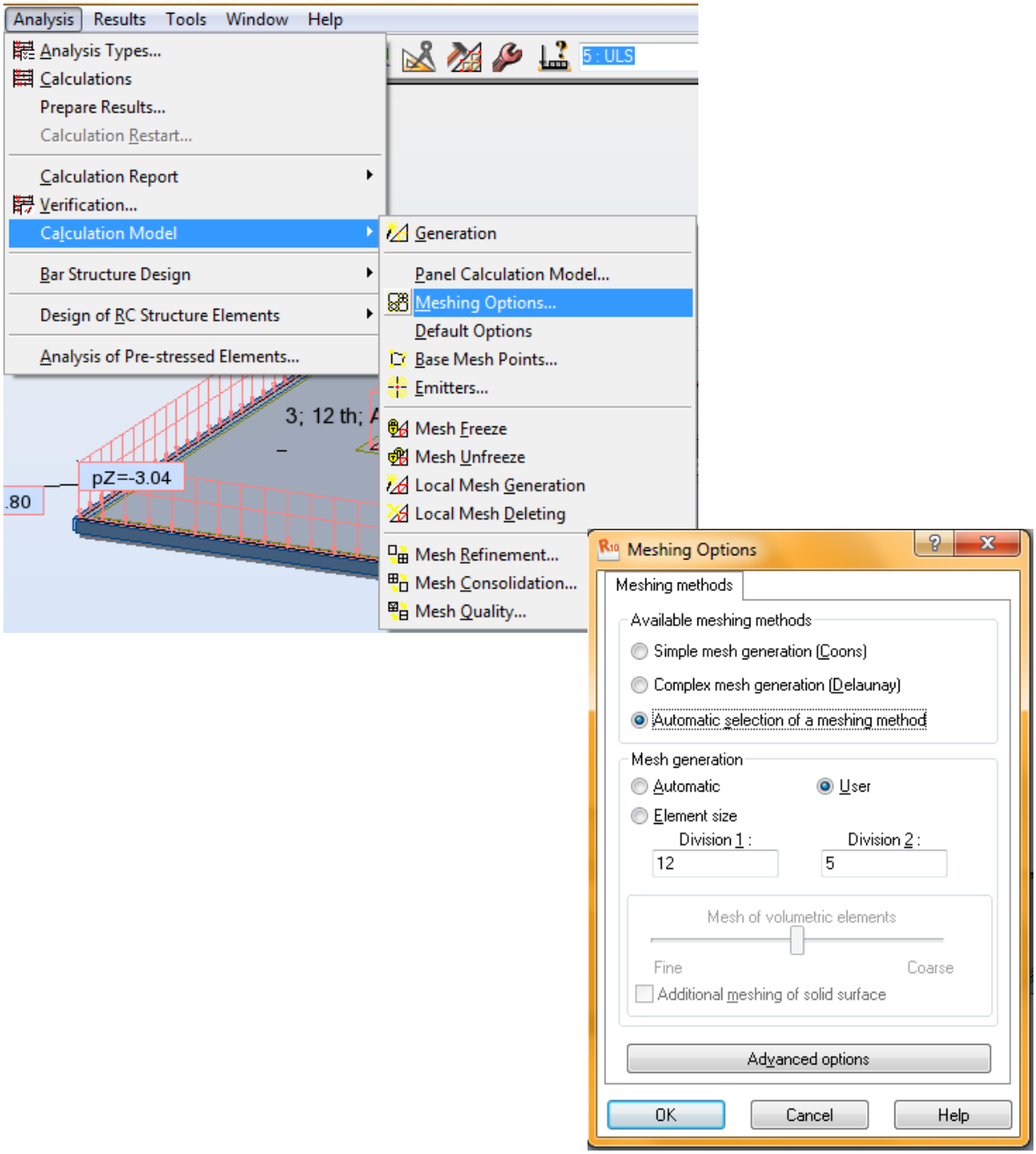


VIII ការ Mesh កំរាលថ្នង់សេ :

- ដំបូងចុចលើរូប  នៅខាងលើអេក្រង់ បន្ទាប់មកចុចលើរូប 



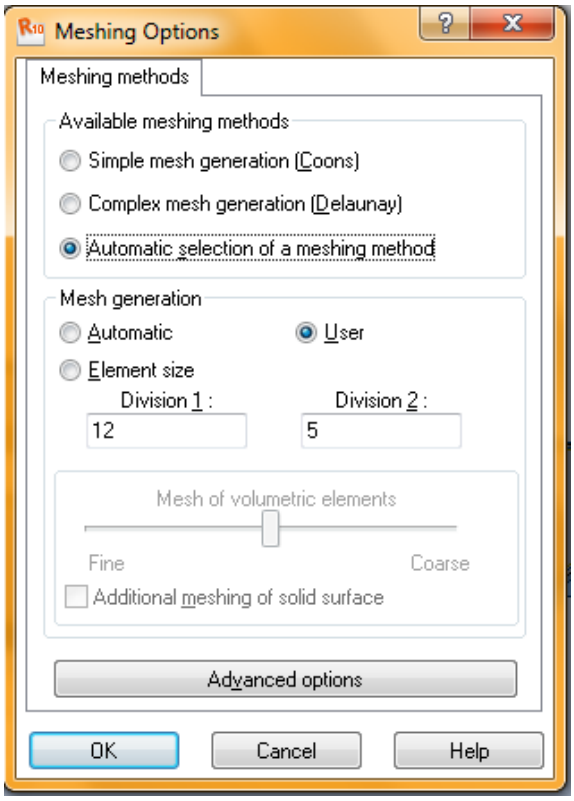
- រចុចលើ Analysis Menu > Calculation Model > Meshing Options…….



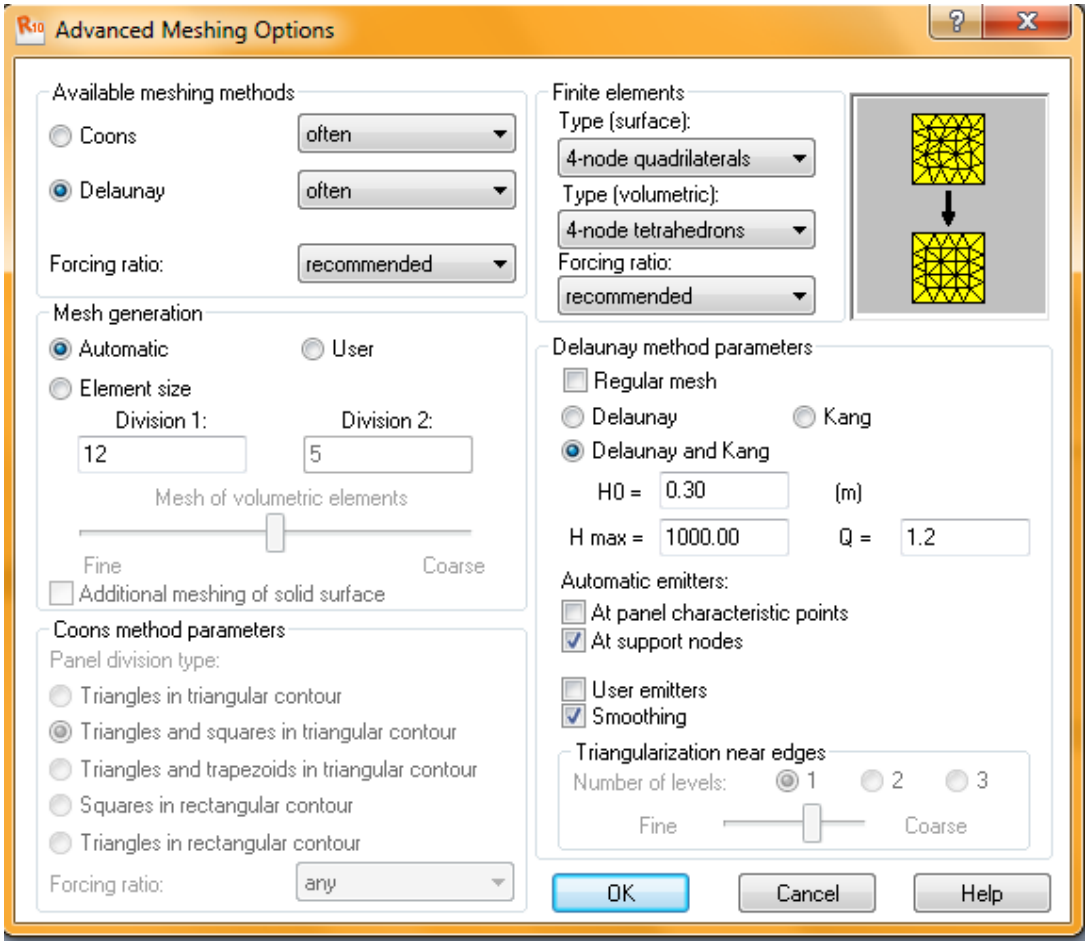
-បន្ទាប់មកដើម្បីបានតារាងខាងក្រោមយើងត្រូវចុចនៅក្នុង

តារាង Meshing Option

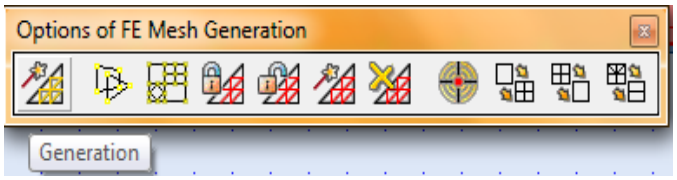
ដោយចុចត្រង់ពាក្យ Advanced Option



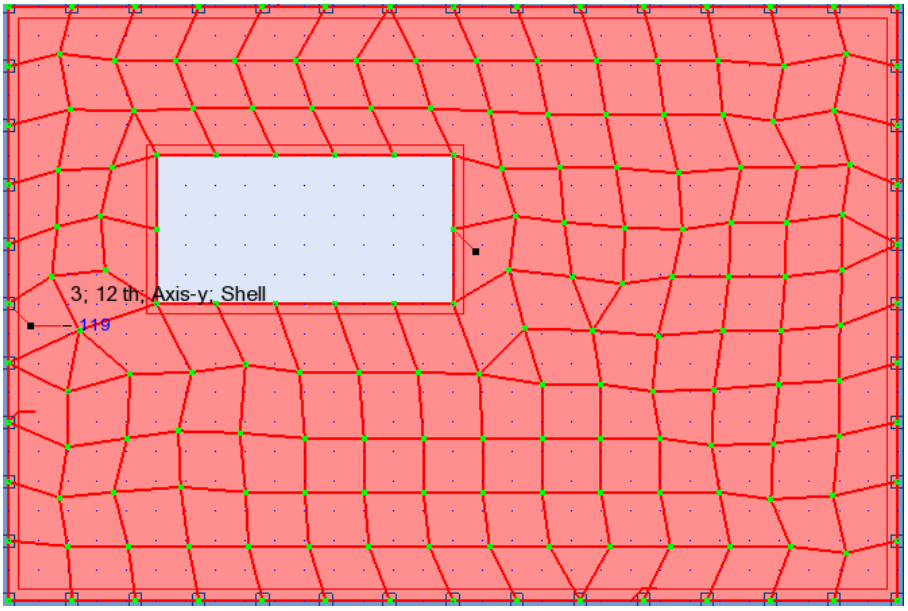
នោះនឹងបានតារាងដូចខាងក្រោម រួចចុច ok



- ធ្វើការ Mesh កំរាលបង្អែសេ
- ដោយចុចលើរូប 

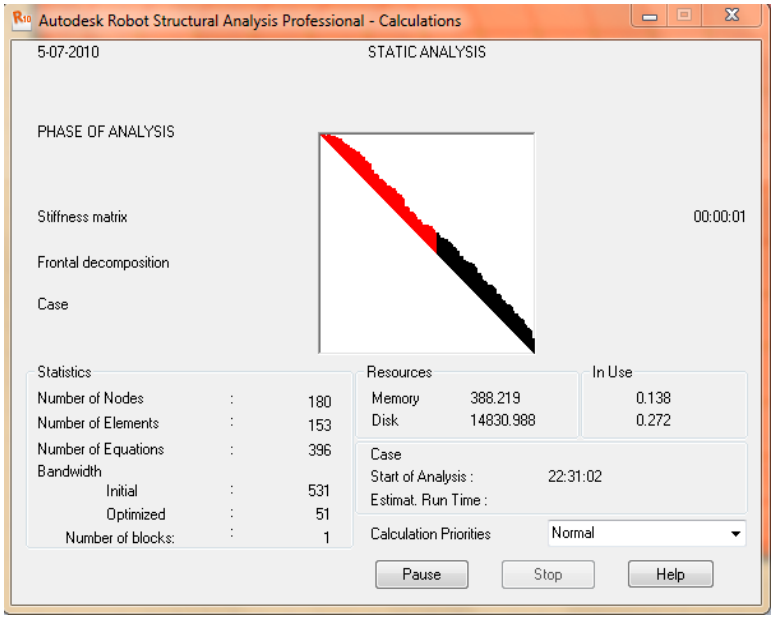


- រូបភាពក្រោយពីធ្វើការ Mesh រួច



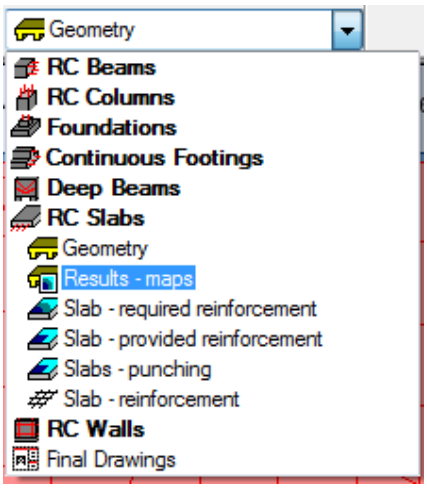
IX ការគណនាកំរាលបង្អែសេ :

-ដំបូងចុចលើរូប Icon  នេះនៅខាងលើអេក្រង់ ដើម្បីគណនារកកំលាំងក្នុងកំរាលបង្អែសេ ។ នោះនឹងចេញផ្ទាំងដូចខាងក្រោម ។ តែបើតារាងនេះចេញមួយភ្លែត ហើយរលត់ទៅវិញនោះមានន័យថា ការធ្វើការគណនារបស់យើងបានល្អ ។ តែផ្ទុយទៅវិញ បើចេញនៅគាំងស្ងៀញ មានន័យថាការគណនារបស់យើងមិនបានល្អ ។

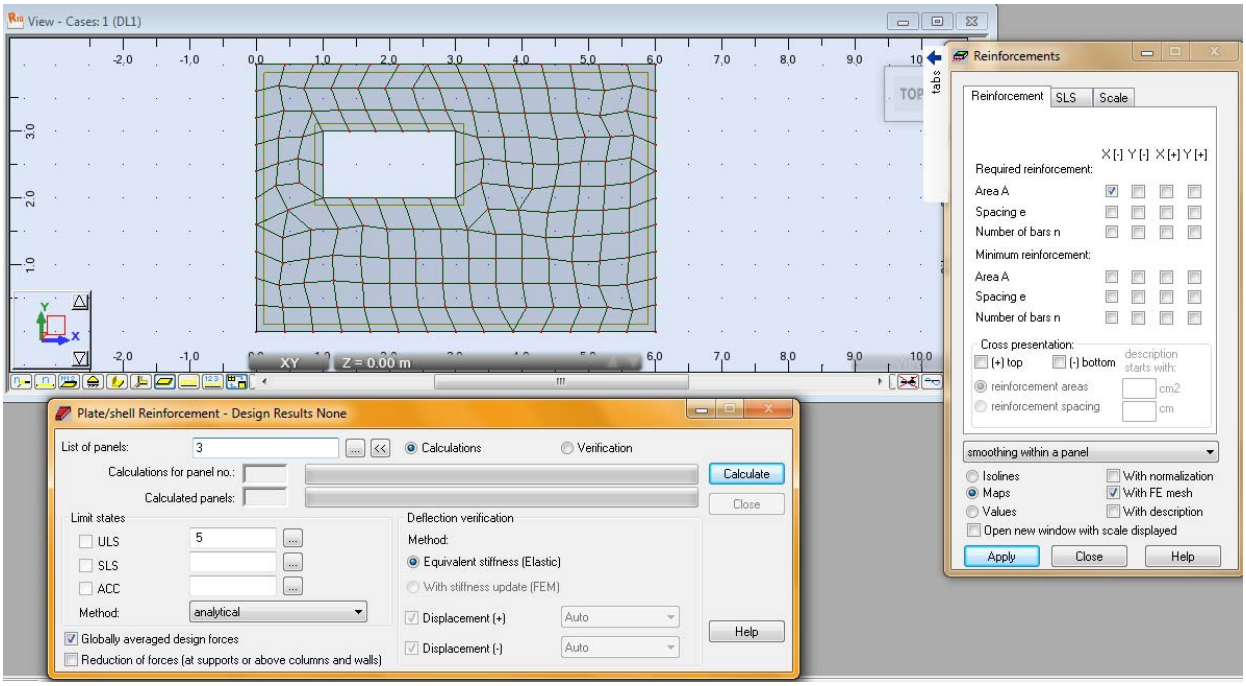


~ ការមើលលទ្ធផល Result – maps

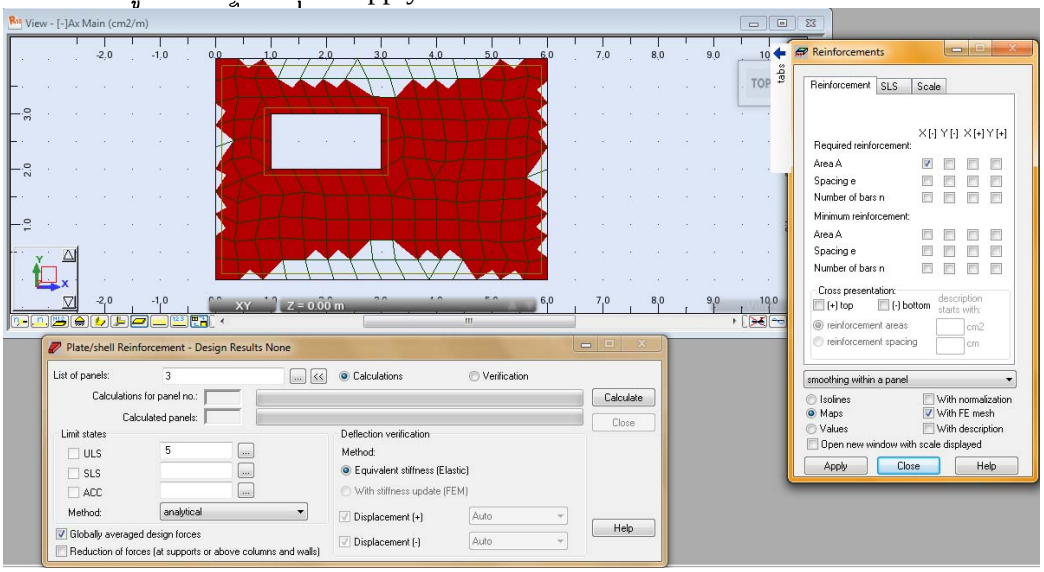
-ចុចលើ Layout រួចរើសយក Result-maps

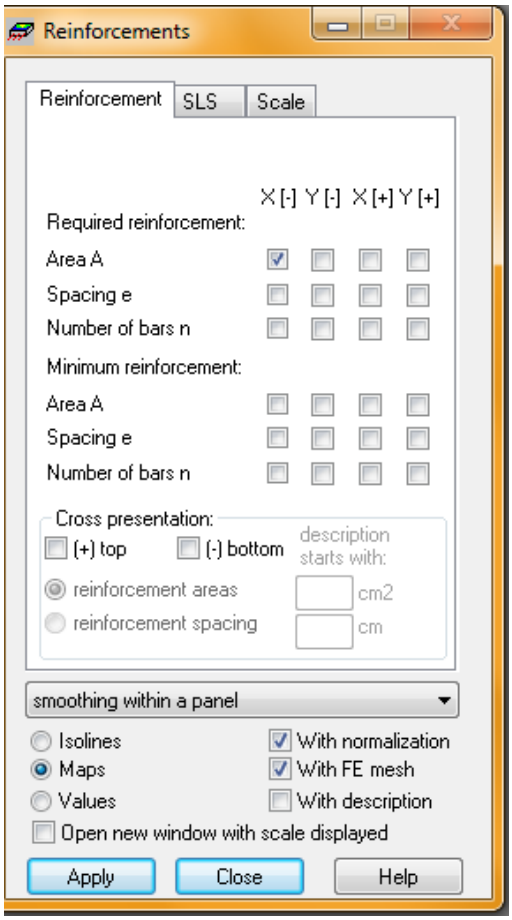


នោះវានឹងចេញតារាងដូចខាងក្រោម :

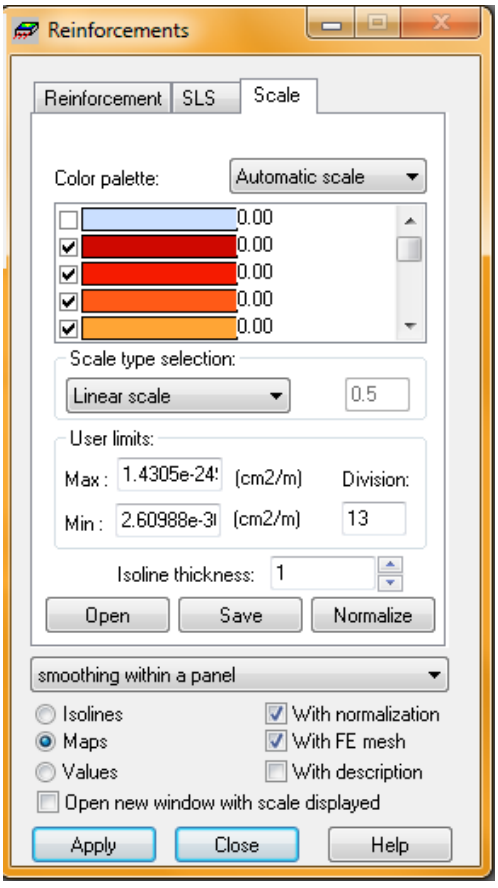


រូបភាពបន្ទាប់ពីចុច Apply ខាងលើ



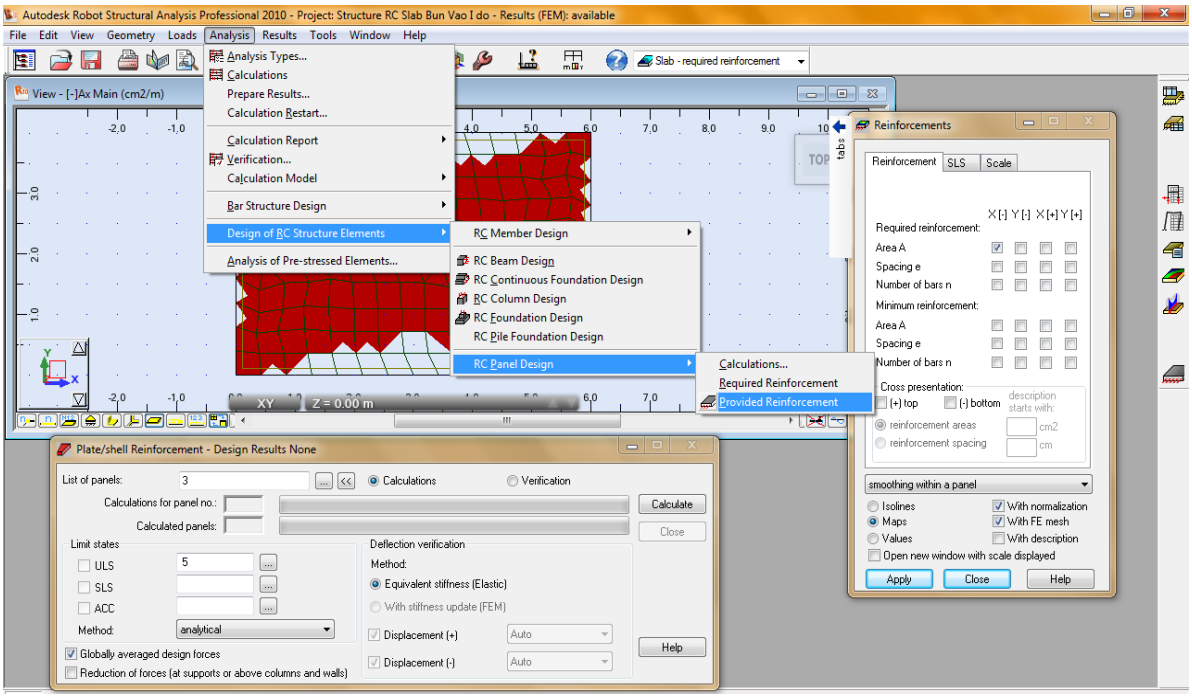

Reinforcements
Reinforcement
SLS
Scale
X [-] Y [-] X [+] Y [+]
Required reinforcement:
Area A
Spacing e
Number of bars n
Minimum reinforcement:
Area A
Spacing e
Number of bars n
Cross presentation:
(+) top
(-) bottom
description starts with:
reinforcement areas
cm2
reinforcement spacing
cm
smoothing within a panel
Isolines
Maps
Values
With normalization
With FE mesh
With description
Open new window with scale displayed
Apply
Close
Help


Reinforcements
Reinforcement
SLS
Scale
X [-] Y [-] X [+] Y [+]
Cracking a
Stiffness factor
Deflection u
smoothing within a panel
Isolines
Maps
Values
With normalization
With FE mesh
With description
Open new window with scale displayed
Apply
Close
Help

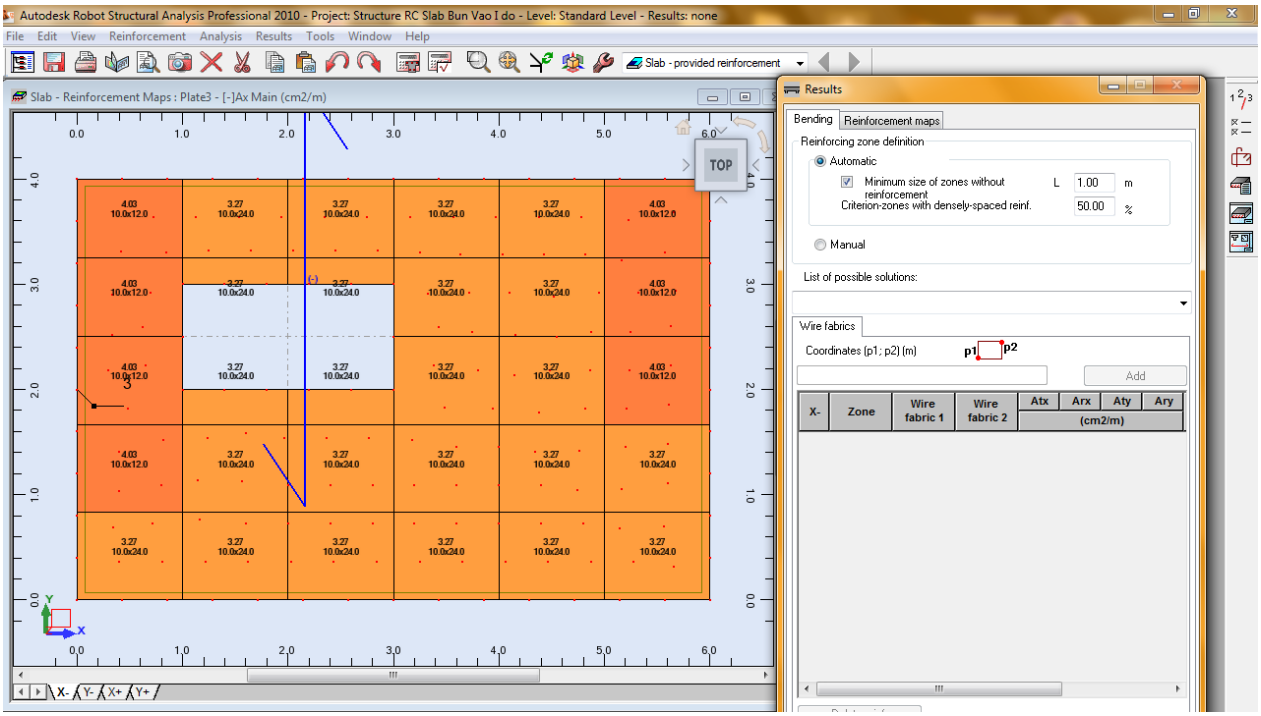

Reinforcements
Reinforcement
SLS
Scale
Color palette:
Automatic scale
0.00
0.00
0.00
0.00
0.00
Scale type selection:
Linear scale
0.5
User limits:
Max :
1.4305e-24
(cm2/m)
Division:
Min :
2.60988e-3
(cm2/m)
13
Isoline thickness:
1
Open
Save
Normalize
smoothing within a panel
Isolines
Maps
Values
With normalization
With FE mesh
With description
Open new window with scale displayed
Apply
Close
Help

X ការកំណត់សសៃចែក :

1- ដំបូងចូល Analysis រួចចូល Design of RC Structure elements រួចចូល RC Panel Design រួចចូល Provided Reinforcement ។ រួចចូល Icon នេះ  នៅខាងស្តាំដៃ ។

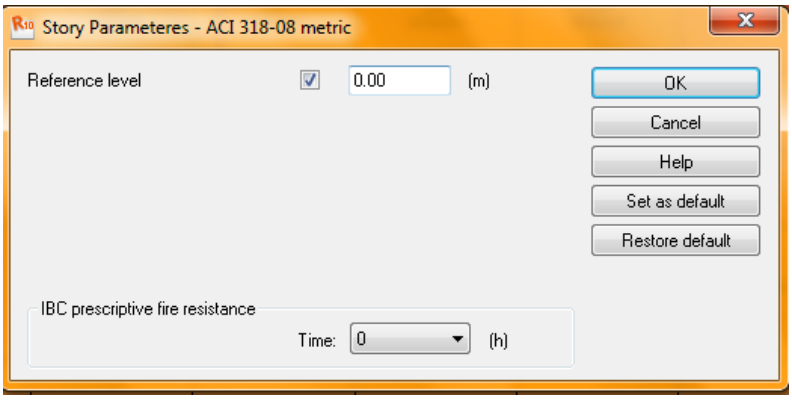


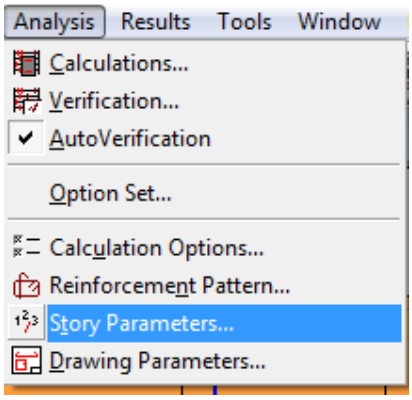
បន្ទាប់ពីចុចលើពាក្យ Provided Reinforcement



2- ការកំណត់ Story Parameters

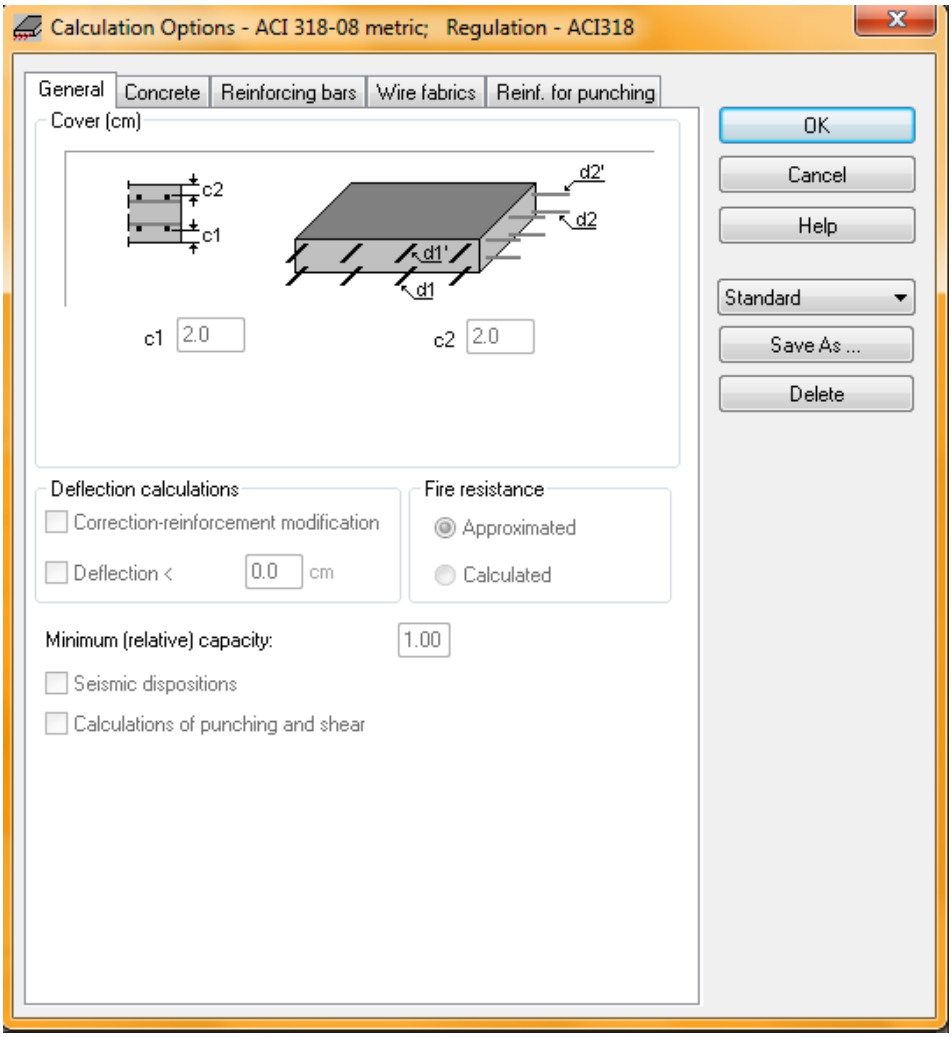
ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំអេក្រង់ រឺ ចុចលើ Analysis Menu > Story Parameters

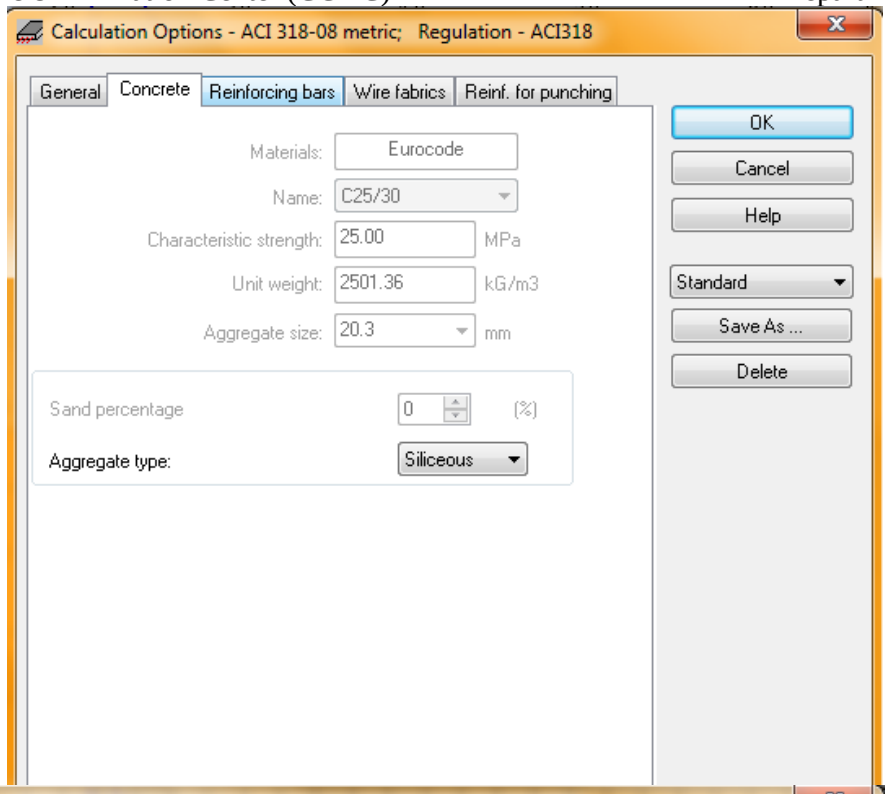


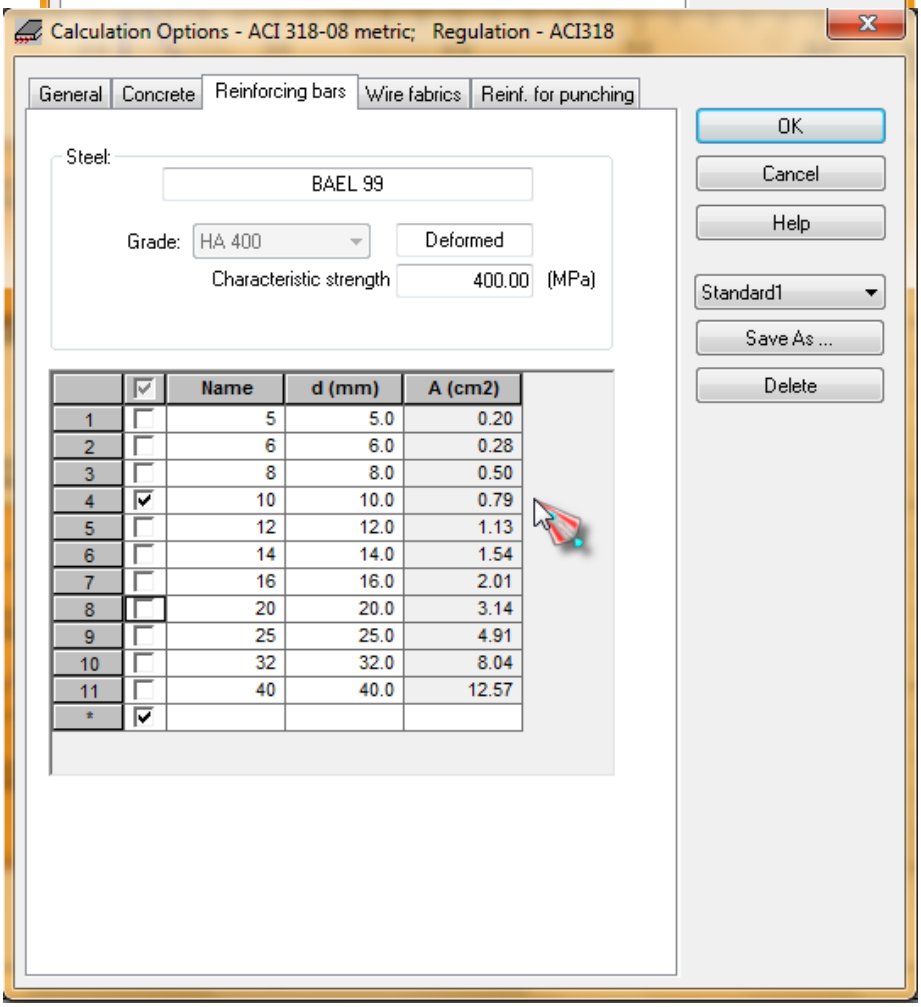


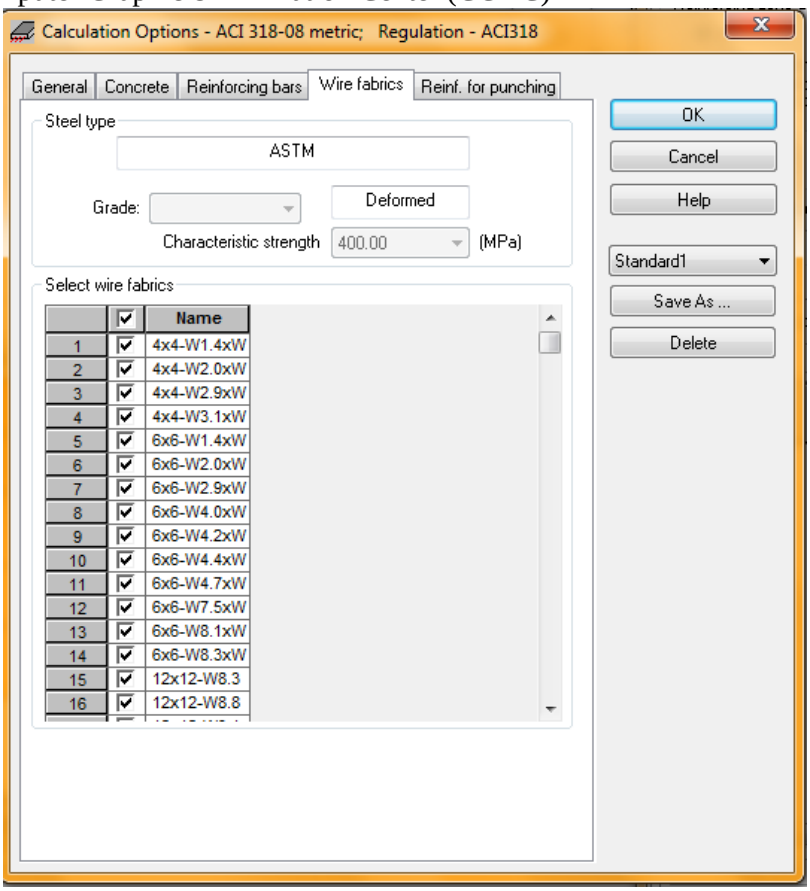
3- ការកំណត់ Calculation Options :

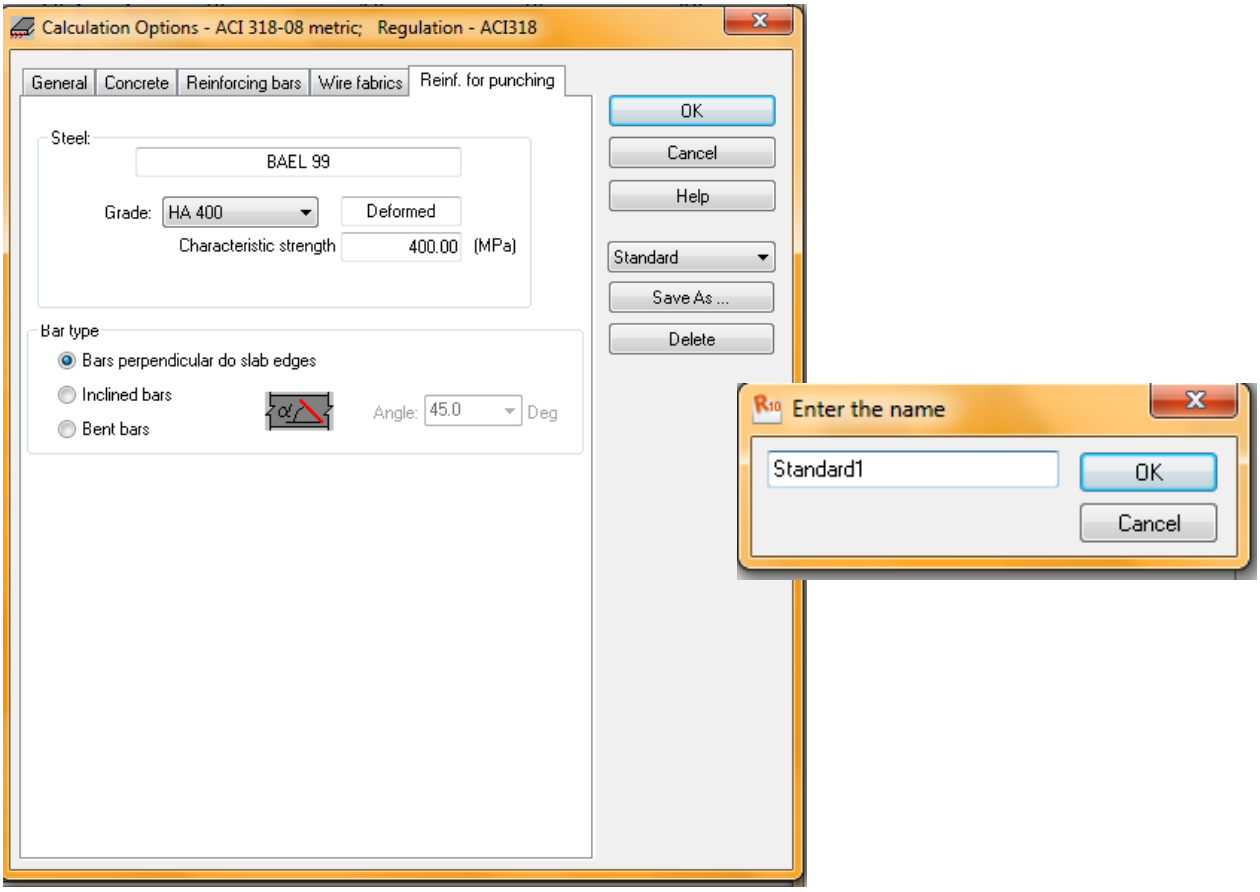
ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំអេក្រង់ រឺ ចុចលើ Analysis Menu > Calculation Option




Calculation Options - ACI 318-08 metric; Regulation - ACI318
General
Concrete
Reinforcing bars
Wire fabrics
Reinf. for punching
Materials:
Eurocode
Name:
C25/30
Characteristic strength:
25.00
MPa
Unit weight:
2501.36
kG/m3
Aggregate size:
20.3
mm
Sand percentage
0
(%)
Aggregate type:
Siliceous
OK
Cancel
Help
Standard
Save As ...
Delete

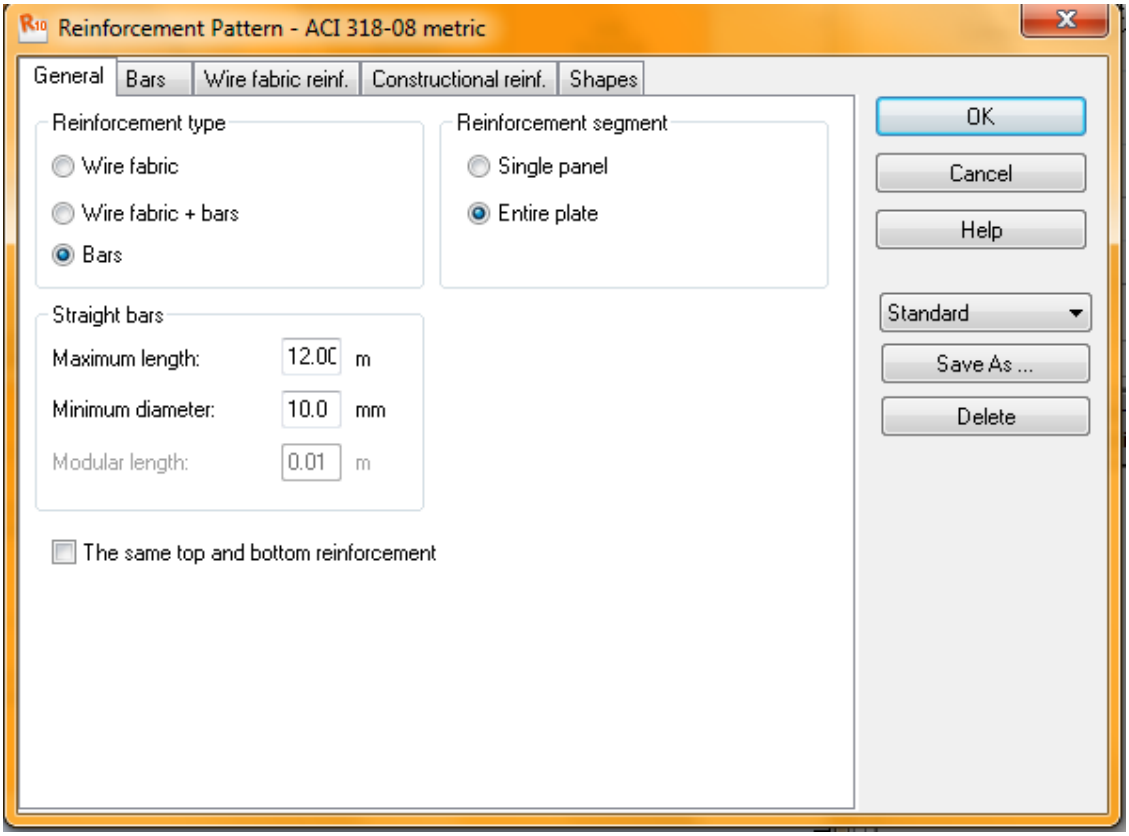

Calculation Options - ACI 318-08 metric; Regulation - ACI318
General
Concrete
Reinforcing bars
Wire fabrics
Reinf. for punching
Steel:
BAEL 99
Grade:
HA 400
Deformed
Characteristic strength
400.00
(MPa)
Name
d (mm)
A (cm2)
1 5 5.0 0.20
2 6 6.0 0.28
3 8 8.0 0.50
4 10 10.0 0.79
5 12 12.0 1.13
6 14 14.0 1.54
7 16 16.0 2.01
8 20 20.0 3.14
9 25 25.0 4.91
10 32 32.0 8.04
11 40 40.0 12.57
OK
Cancel
Help
Standard1
Save As ...
Delete

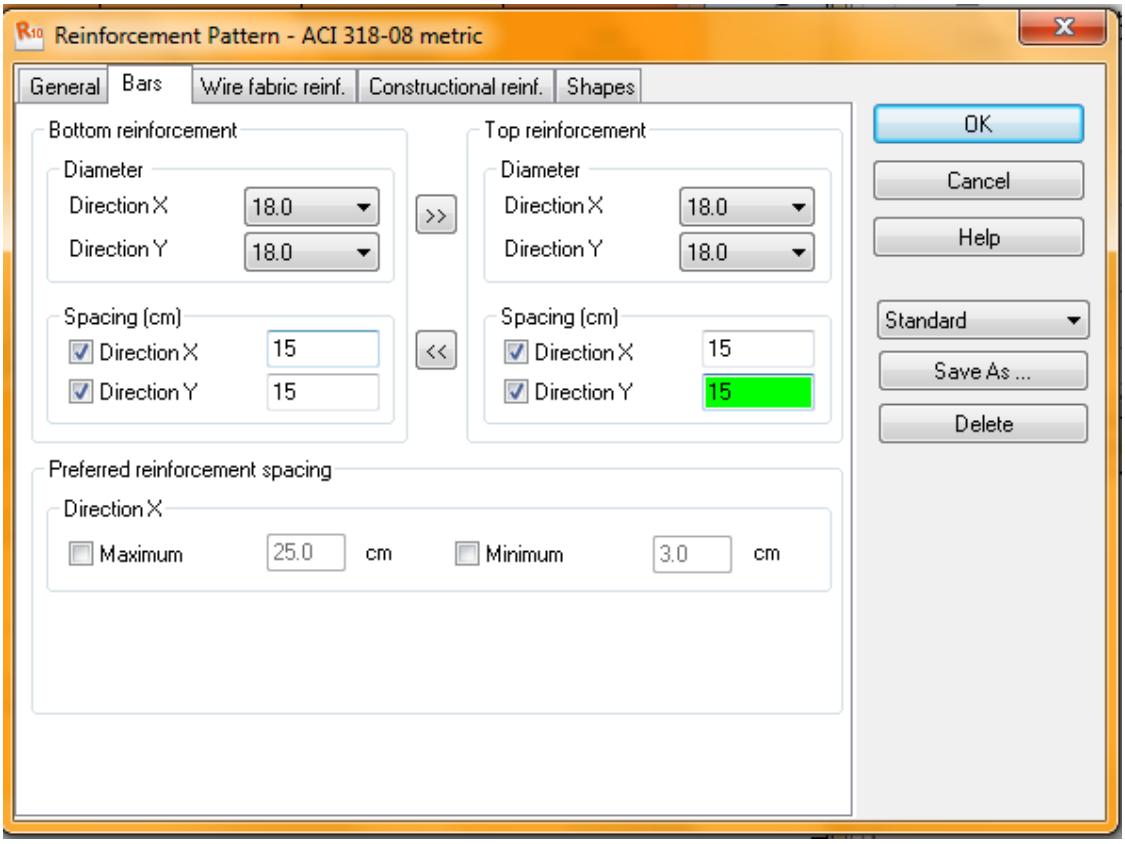


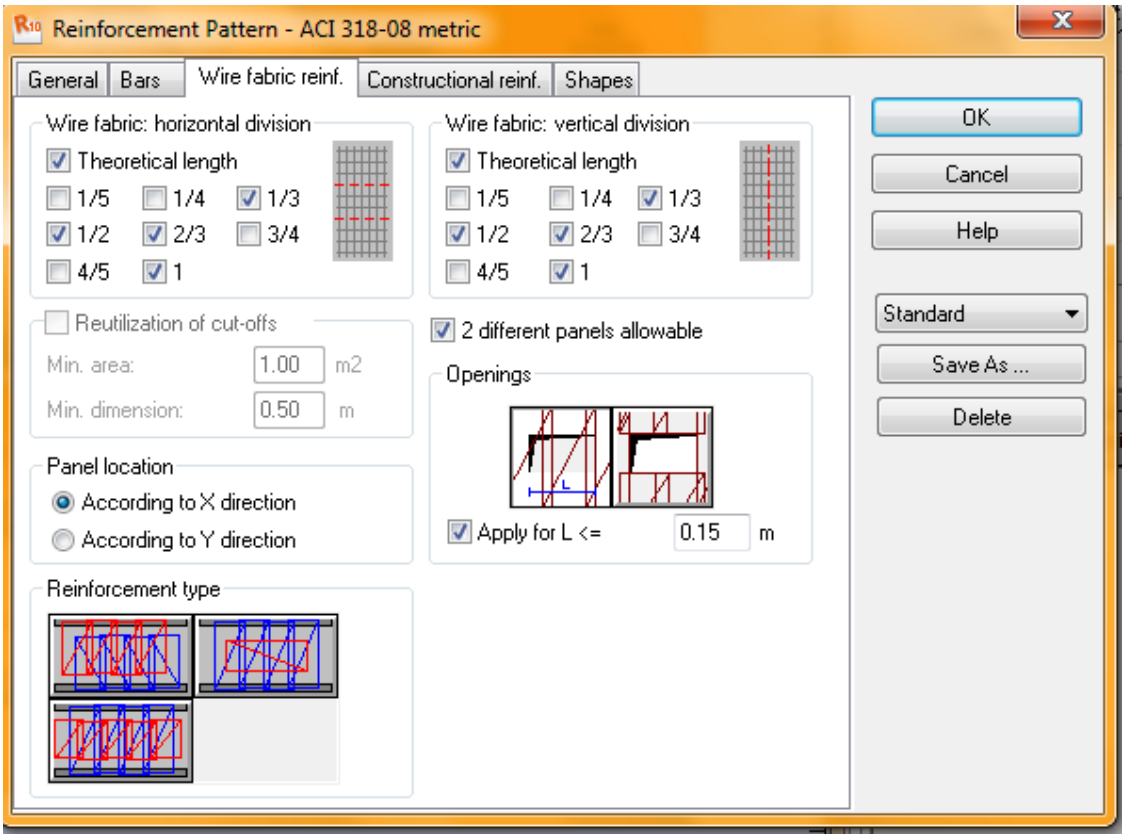


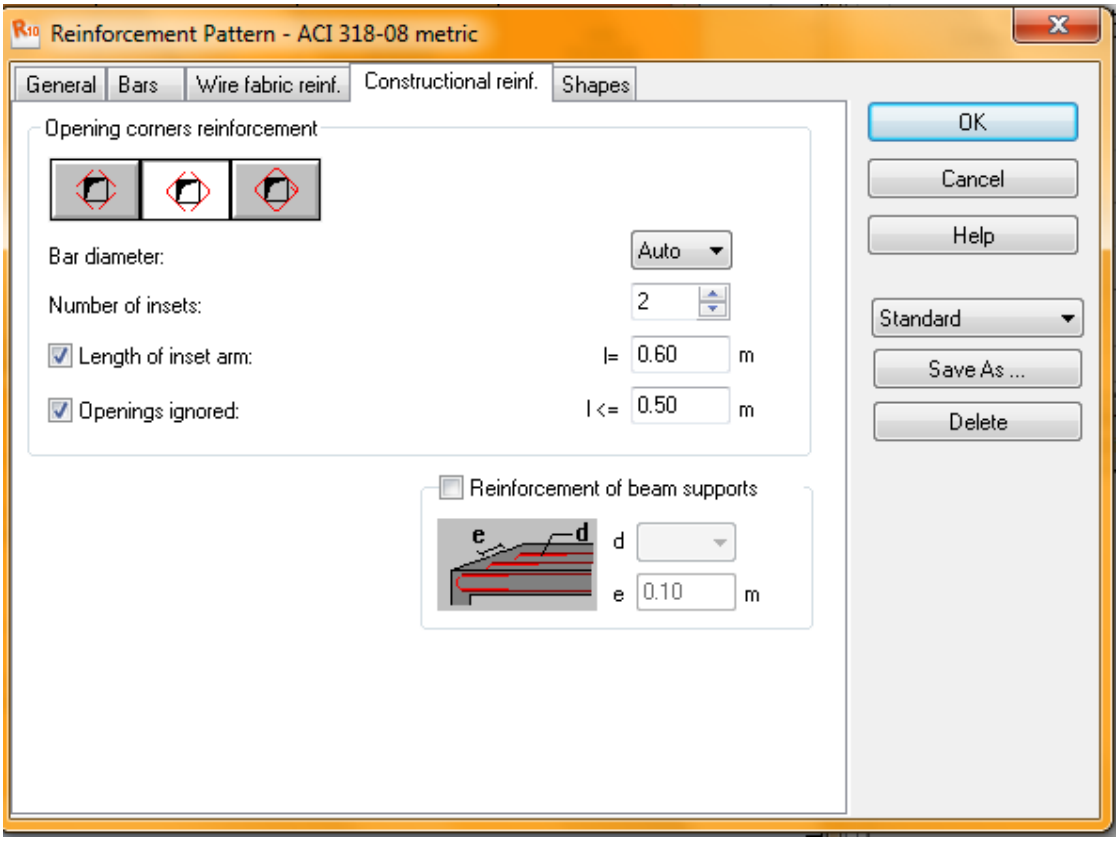
4- ការកំណត់ Reinforcement Pattern :

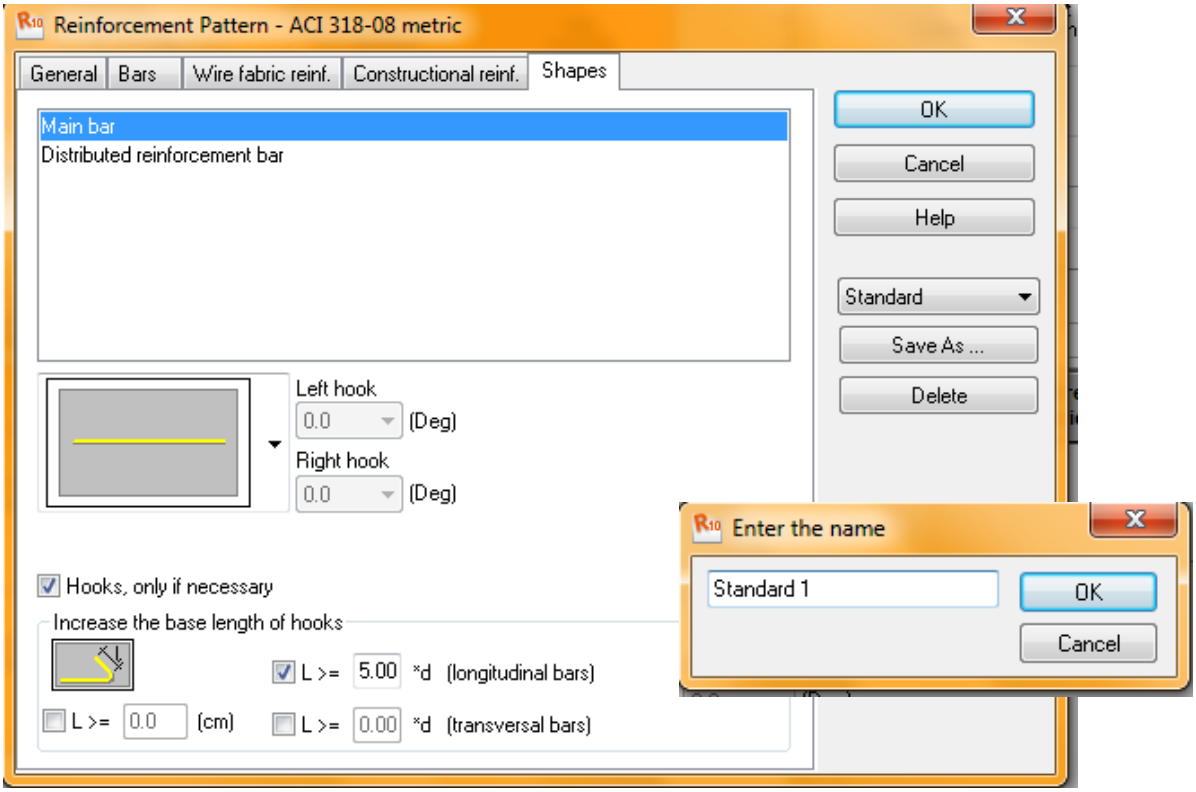
ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំអេក្រង់ រឺ ចុចលើ Analysis Menu > Reinforcement Option





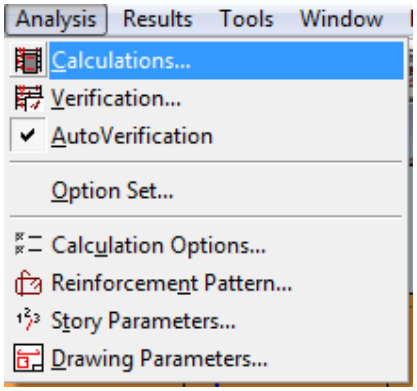

Reinforcement Pattern - ACI 318-08 metric
General
Bars
Wire fabric reinf.
Constructional reinf.
Shapes
Wire fabric: horizontal division
Theoretical length
1/5
1/4
1/3
1/2
2/3
3/4
4/5
1
Wire fabric: vertical division
Theoretical length
1/5
1/4
1/3
1/2
2/3
3/4
4/5
1
Reutilization of cut-offs
Min. area:
1.00
m2
Min. dimension:
0.50
m
2 different panels allowable
Openings
Apply for L <=
0.15
m
Panel location
According to X direction
According to Y direction
Reinforcement type
OK
Cancel
Help
Standard
Save As ...
Delete


Reinforcement Pattern - ACI 318-08 metric
General
Bars
Wire fabric reinf.
Constructional reinf.
Shapes
Opening corners reinforcement
Bar diameter:
Auto
Number of insets:
2
Length of inset arm:
l=
0.60
m
Openings ignored:
l <=
0.50
m
Reinforcement of beam supports
e
d
d
e
0.10
m
OK
Cancel
Help
Standard
Save As ...
Delete

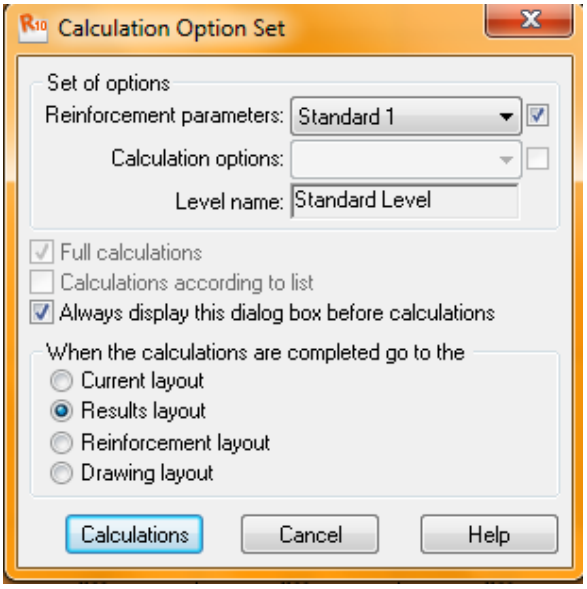


XI- ចាប់ផ្តើមគណនារកសរសៃដែក Start Calculate Provided Reinforcement :

ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំអេក្រង់ រឺ ចុចលើ Analysis Menu > Reinforcement Option

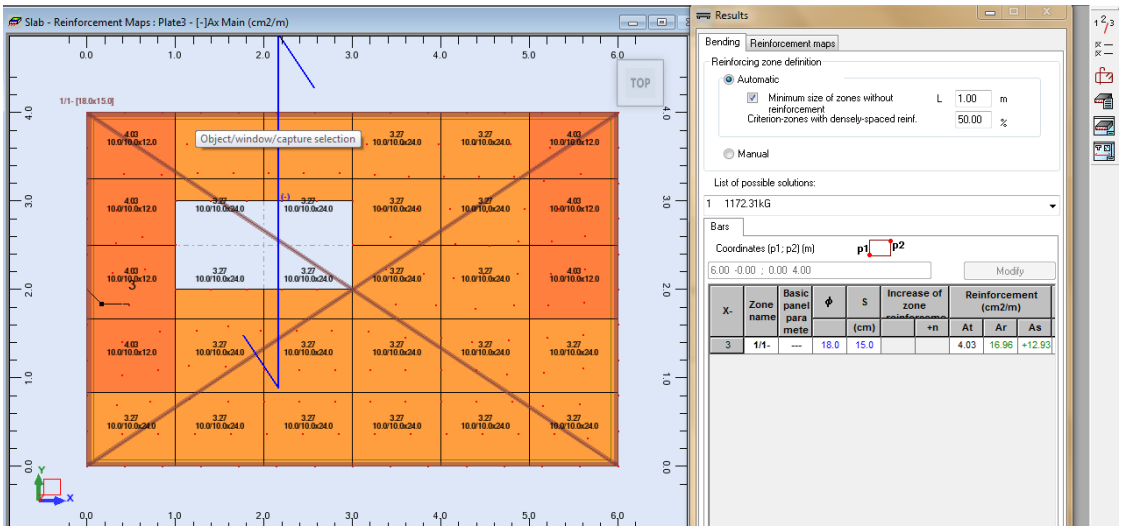


ពេលនោះវានឹងបង្ហាញផ្ទាំង Calculation Option Set



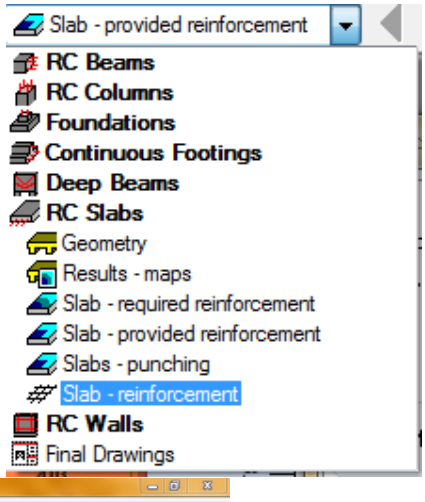
ជ្រើសរើសឈ្មោះត្រង់ Reinforcement parameters និង Calculation options ដែលយើងបានកំណត់រួចមក ។ រួចចុច Calculations .

បន្ទាប់ពីចុច Calculation រួច ។

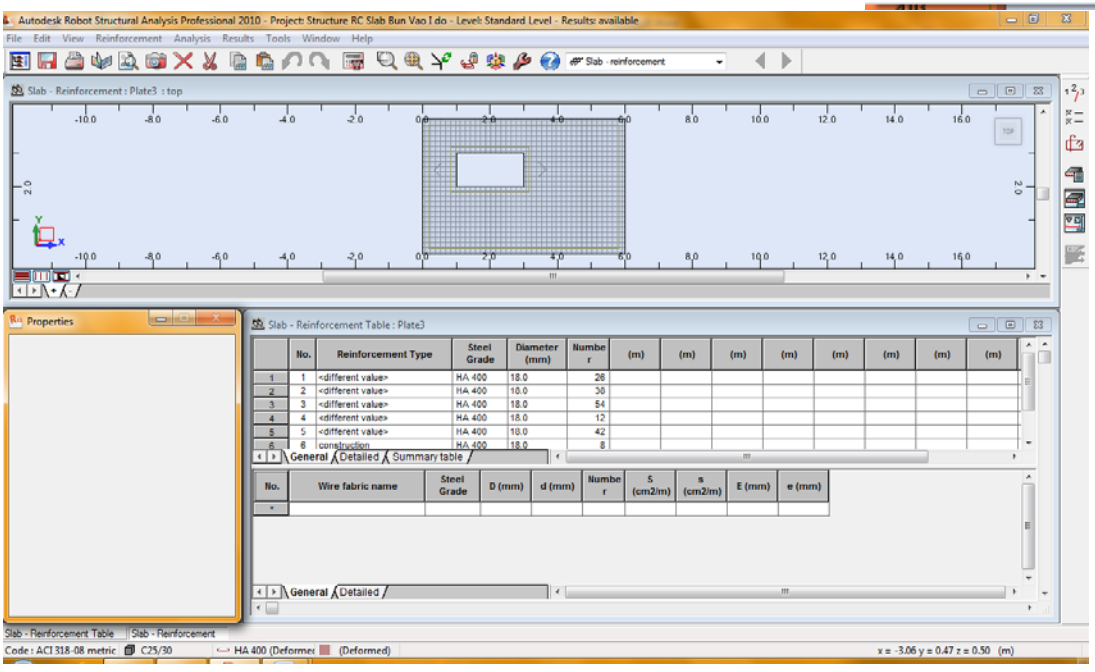


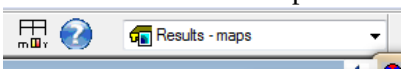
XII ការត្រួតពិនិត្យមើលការដាក់សរសៃដែក Slab-Reinforcement

-ដំបូងចុចលើ Layout រើសយក Slab-reinforcement

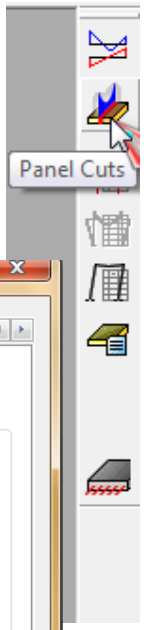


បង្ហាញដូចរូប

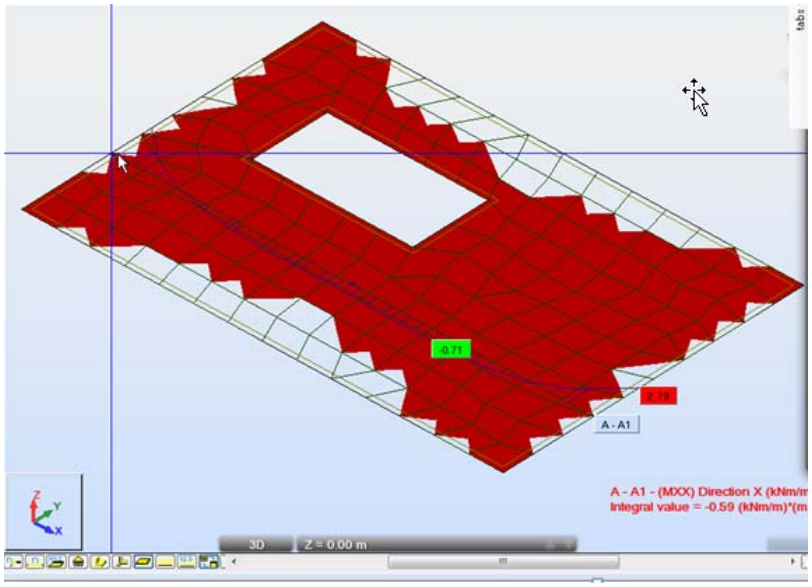


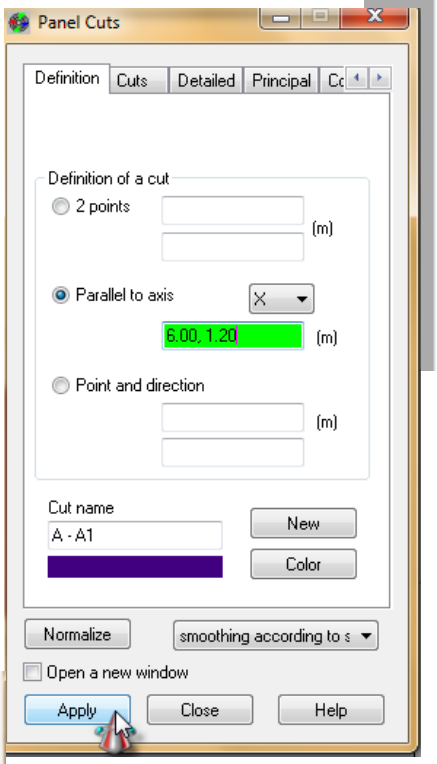
ការបង្ហាញមើលម៉ូម៉ង់ : ដំបូងត្រូវកែខាងលើយក Results-maps 

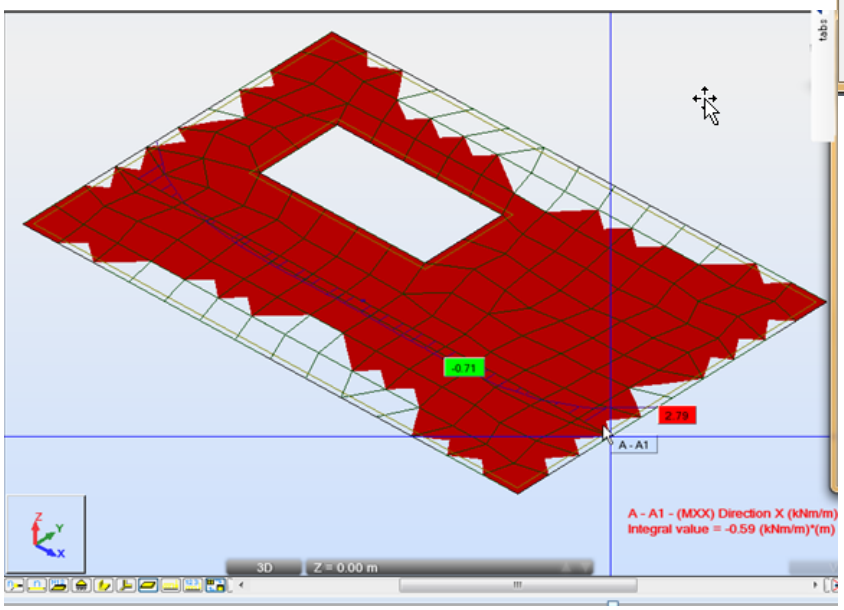
រួចយក Mouse ទៅចុចលើ Icon  នេះនៅខាងស្តាំអេក្រង់ នឹងចេញផ្ទាំងដូចខាងក្រោម



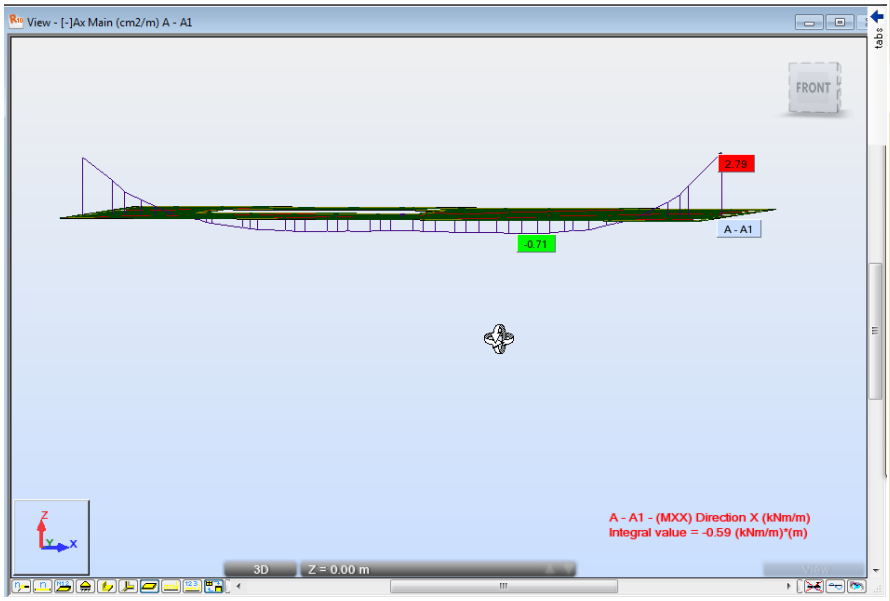
រួចចូល Definition ហើយចុចយក Parallel to axis X រួចដាក់ Mouse នៅក្នុងប្រអប់ក្រោមវា រួចចុច Apply រួចយកមកចុចនៅលើ Slab តាមទិស X ។ ដូចរូប

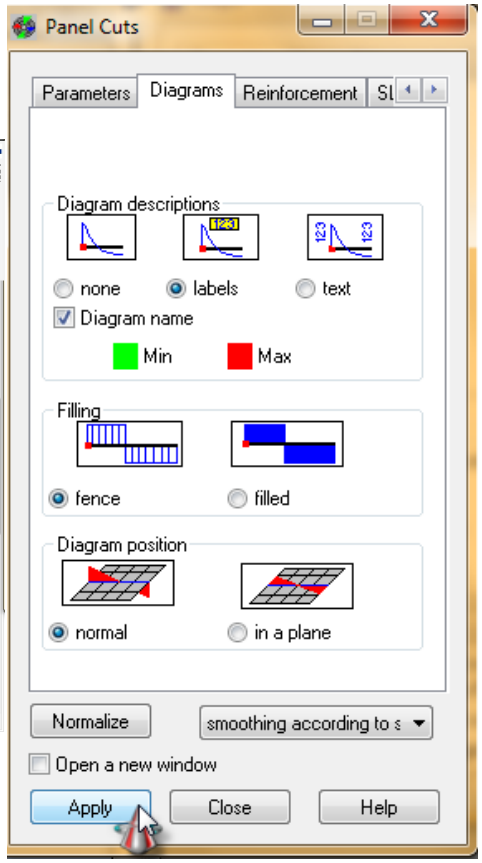




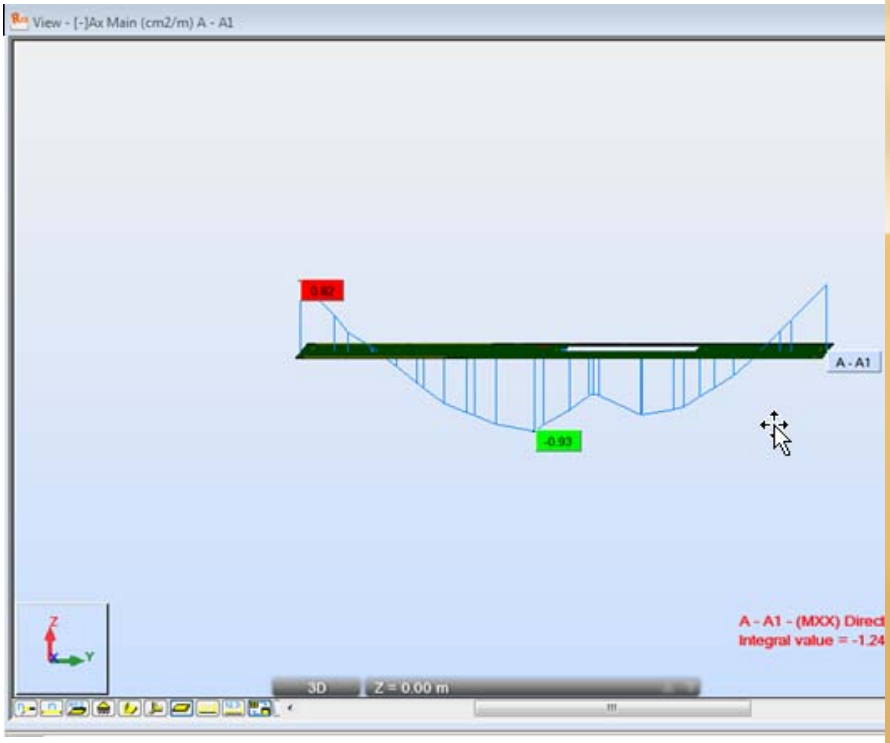


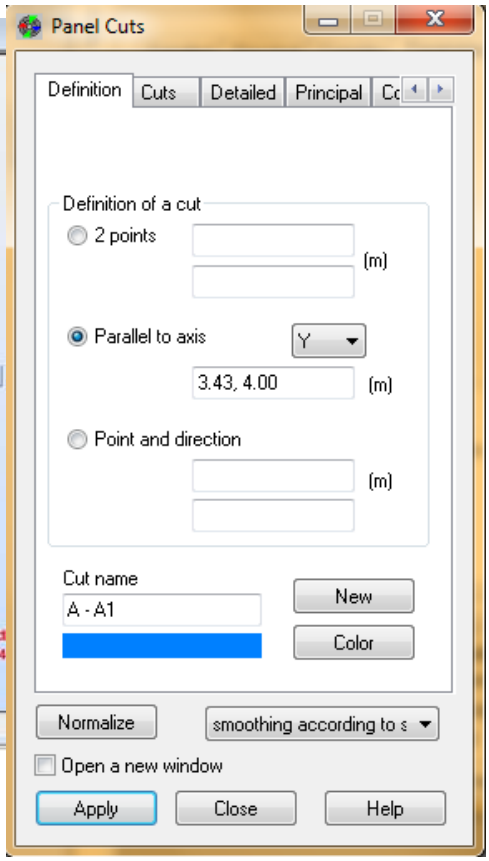
បន្ទាប់មកចូល Diagrams ចុចត្រង់ Labels និង Normal រួចចុច Apply ដើម្បីមើល Diagrams វា ។ ដូចរូប





រួចចូលតាមទិស Y ដូចរូប

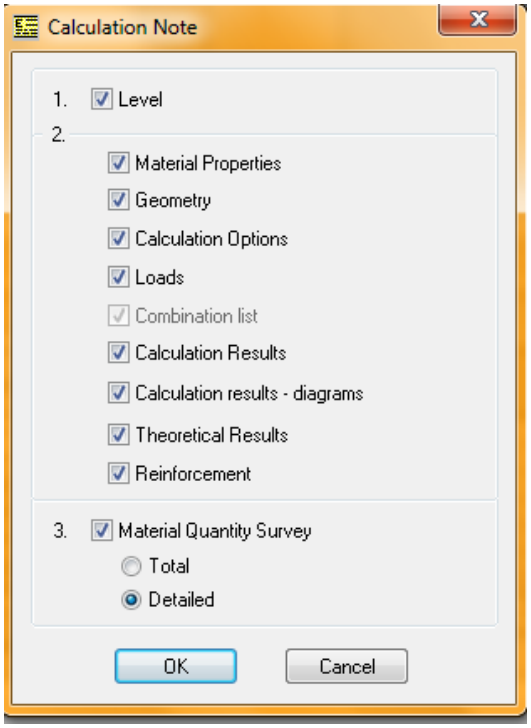


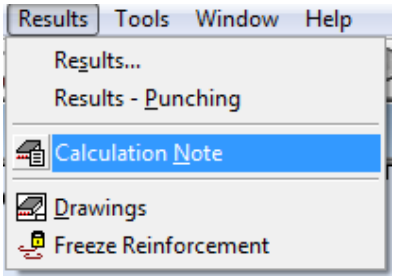


XIII Calculation – Note

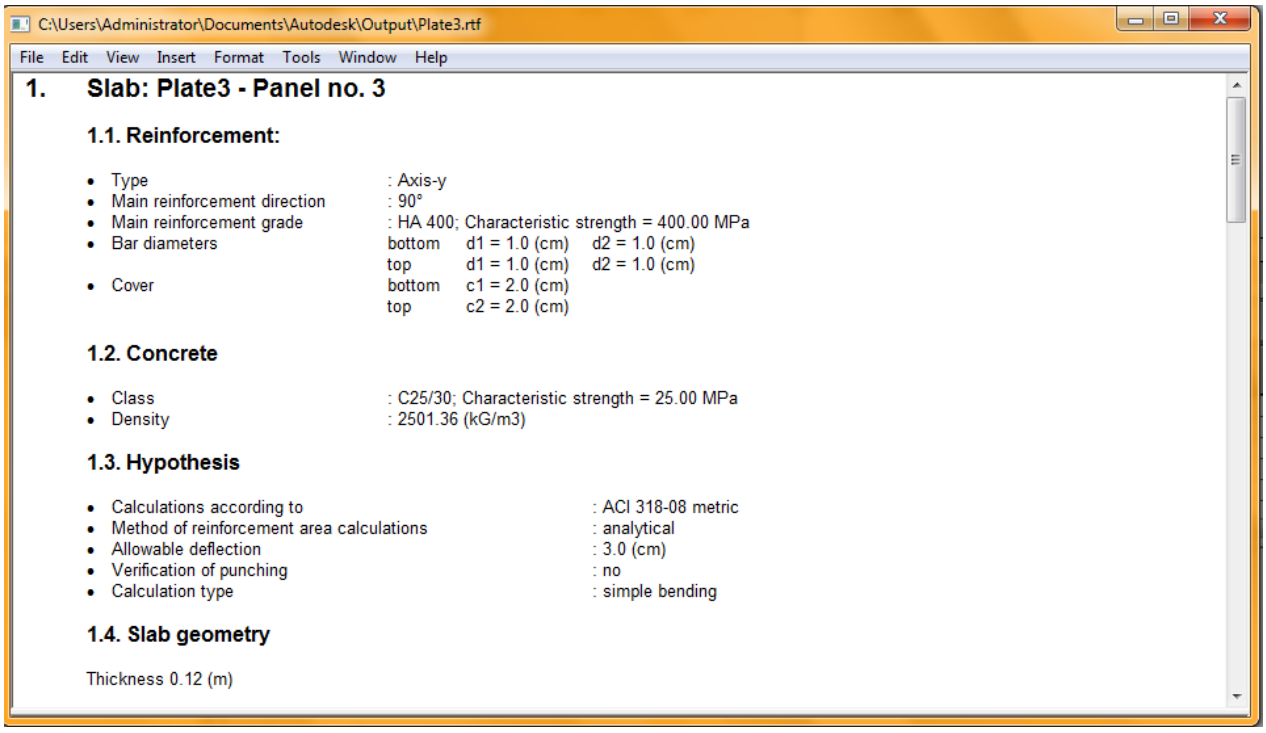
www.yerngshare.blogspot.com

ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំអេក្រង់ រឺ ចុចលើ Analysis Menu > Calculation– Note



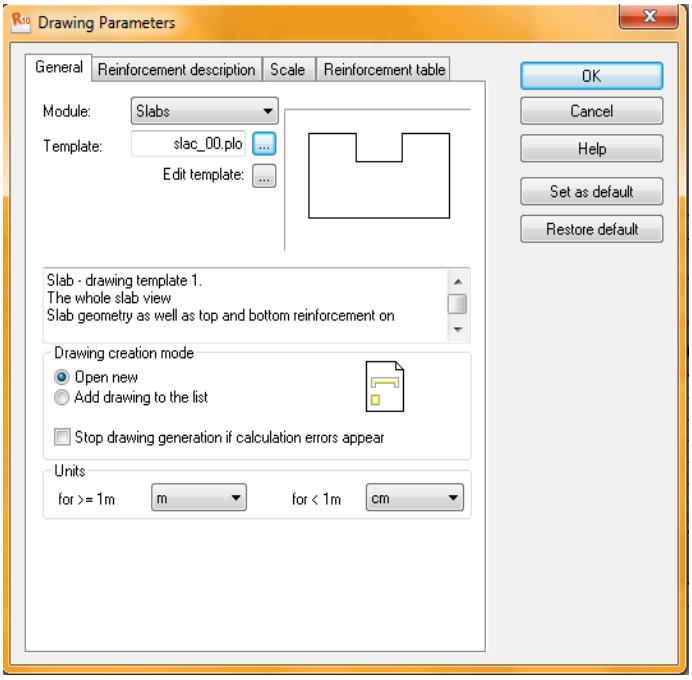


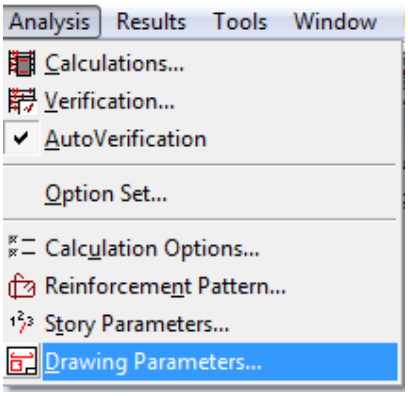
បន្ទាប់ពីចុច OK វានឹងចេញផ្ទាំងពត៌មានដូចខាងក្រោម

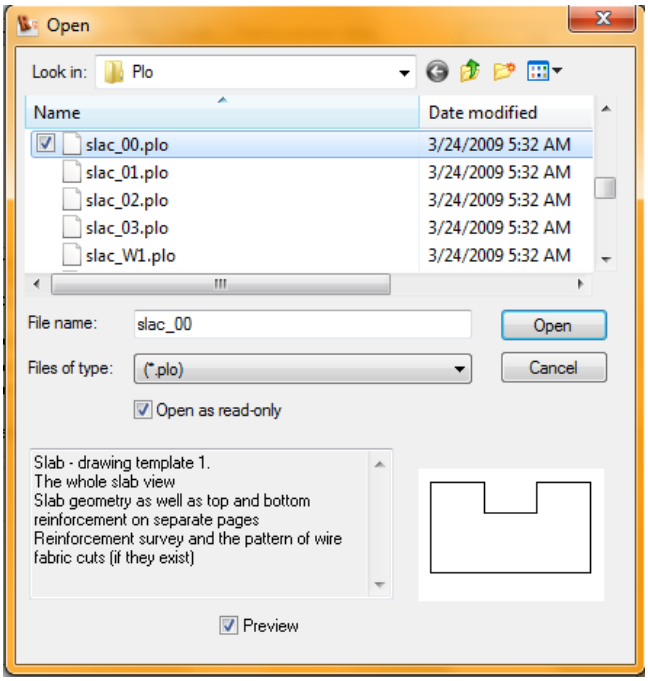


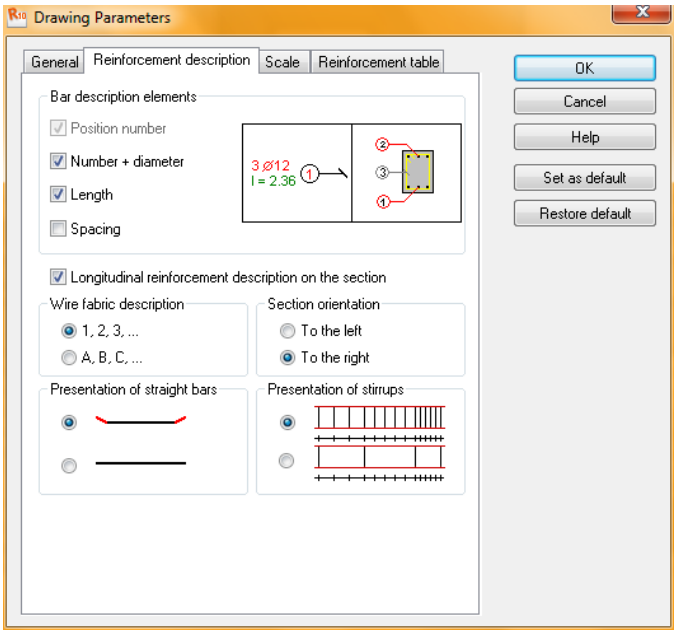
XIV Drawing Parameters

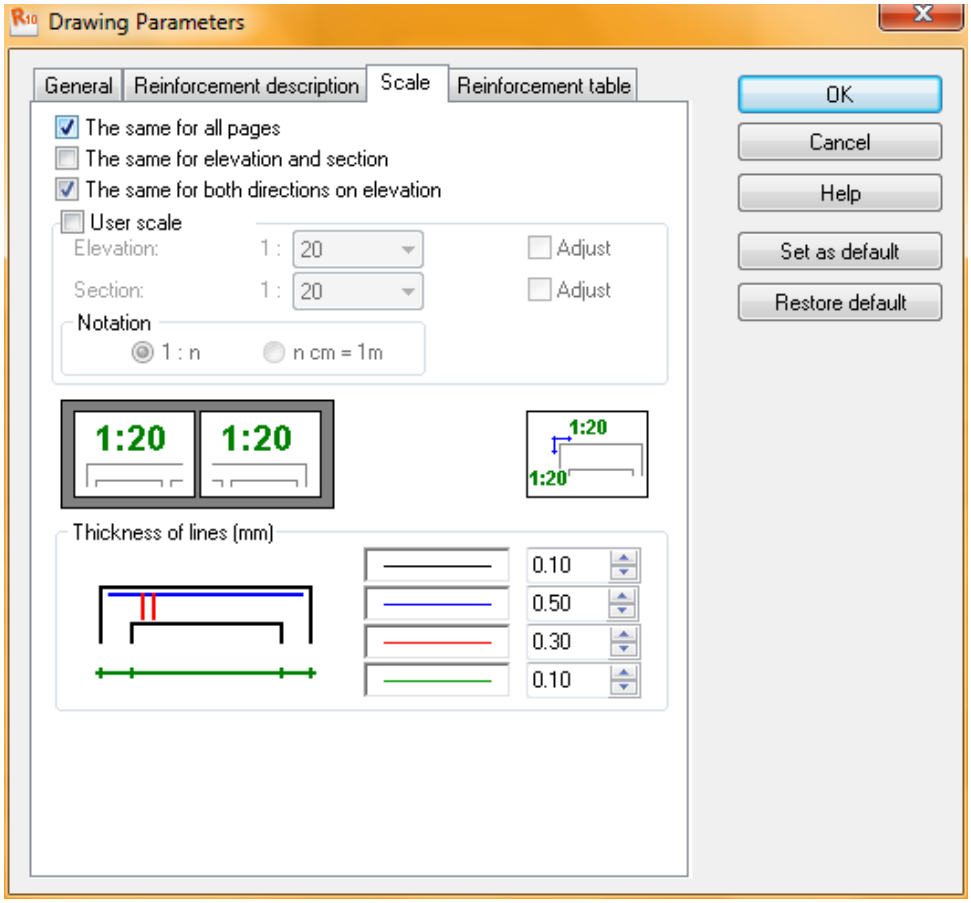
ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំអេក្រង់ រឺ ចុចលើ Analysis Menu > Drawing Parameters

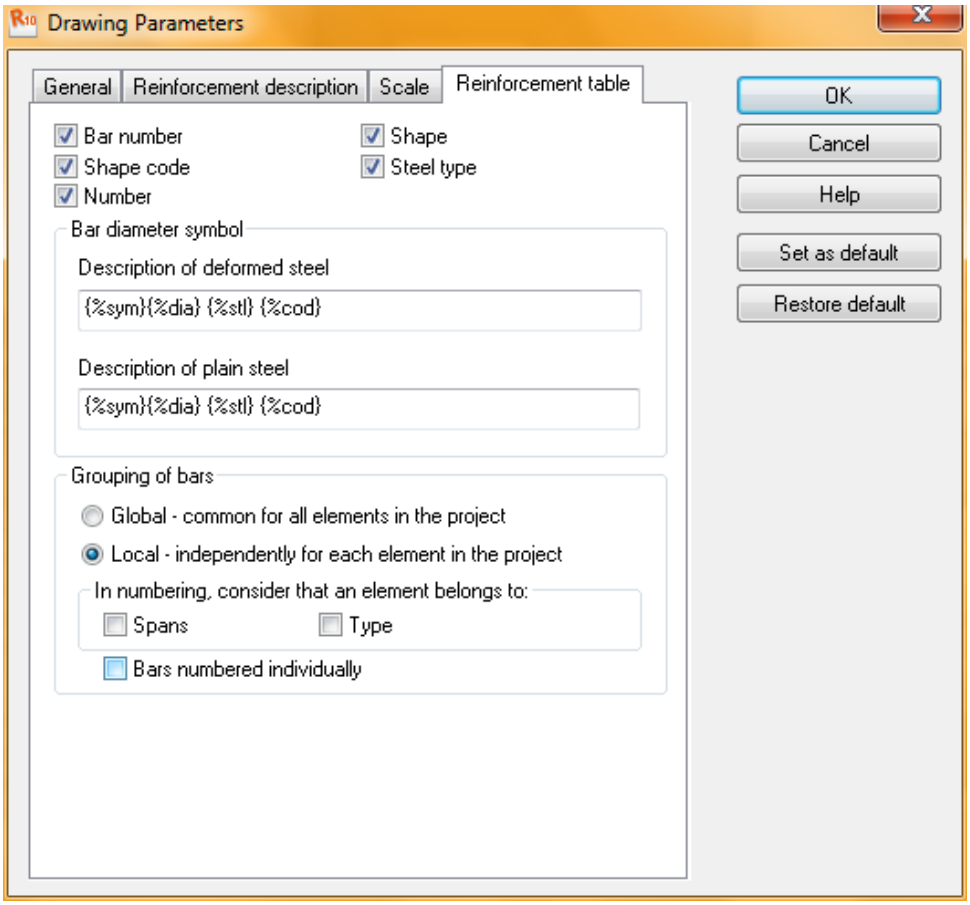






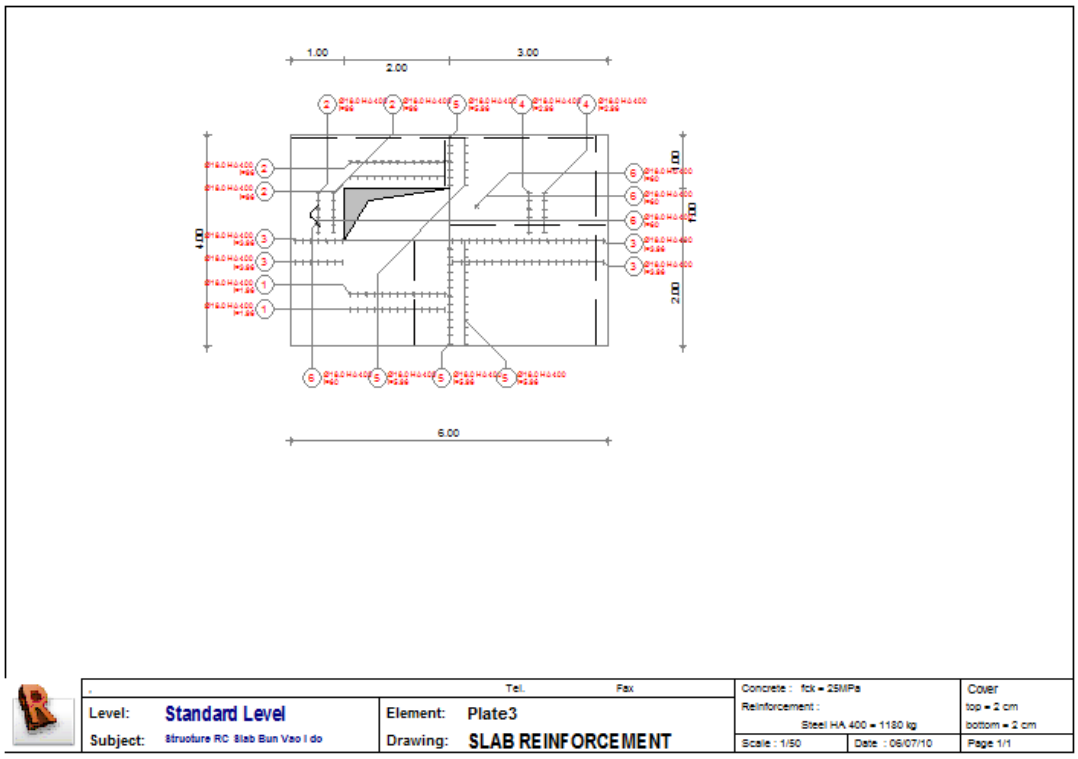


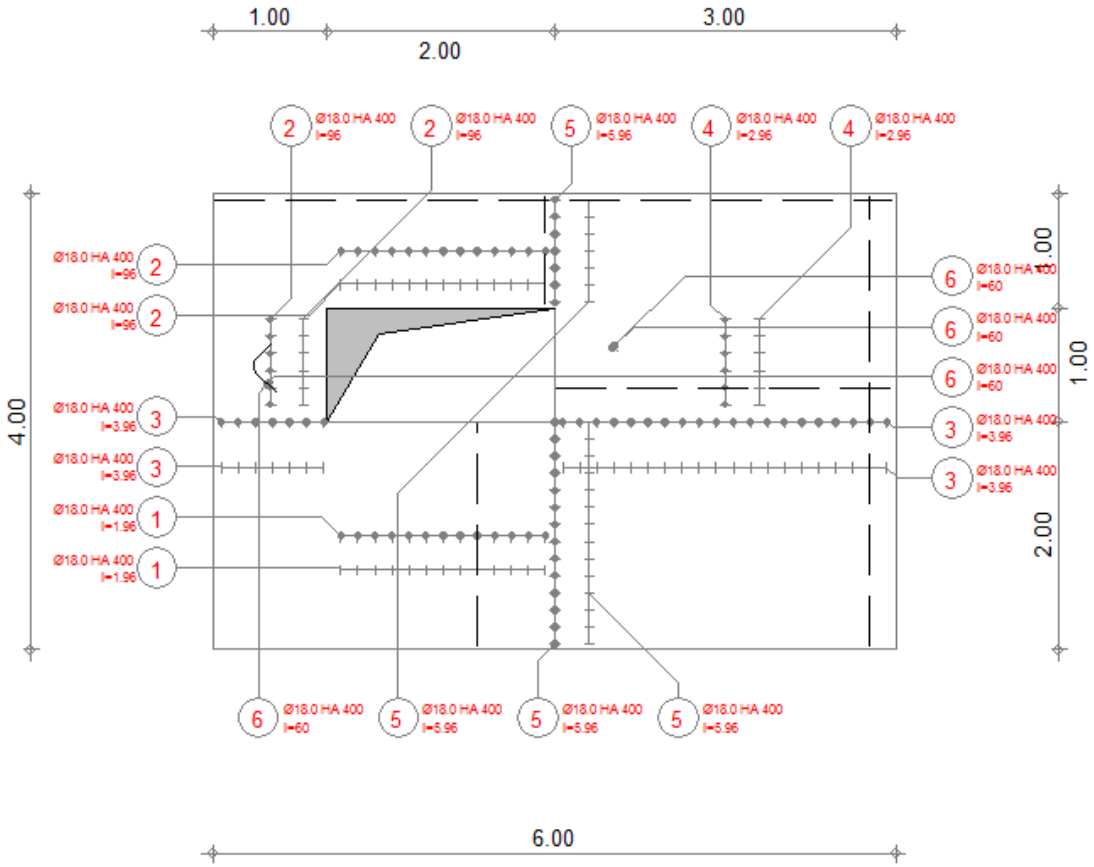

Drawing Parameters
General
Reinforcement description
Scale
Reinforcement table
The same for all pages
The same for elevation and section
The same for both directions on elevation
User scale
Elevation:
1 :
20
Adjust
Section:
1 :
20
Adjust
Notation
1 : n
n cm = 1m
1:20
1:20
1:20
1:20
Thickness of lines (mm)
0.10
0.50
0.30
0.10
OK
Cancel
Help
Set as default
Restore default


Drawing Parameters
General
Reinforcement description
Scale
Reinforcement table
Bar number
Shape code
Number
Shape
Steel type
Bar diameter symbol
Description of deformed steel
{%sym}{%dia} {%stl} {%cod}
Description of plain steel
{%sym}{%dia} {%stl} {%cod}
Grouping of bars
Global - common for all elements in the project
Local - independently for each element in the project
In numbering, consider that an element belongs to:
Spans
Type
Bars numbered individually
OK
Cancel
Help
Set as default
Restore default

XV មើលប្លង់ Drawing

ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំអេក្រង់ រឺ ចុចលើ Result Menu > Drawings



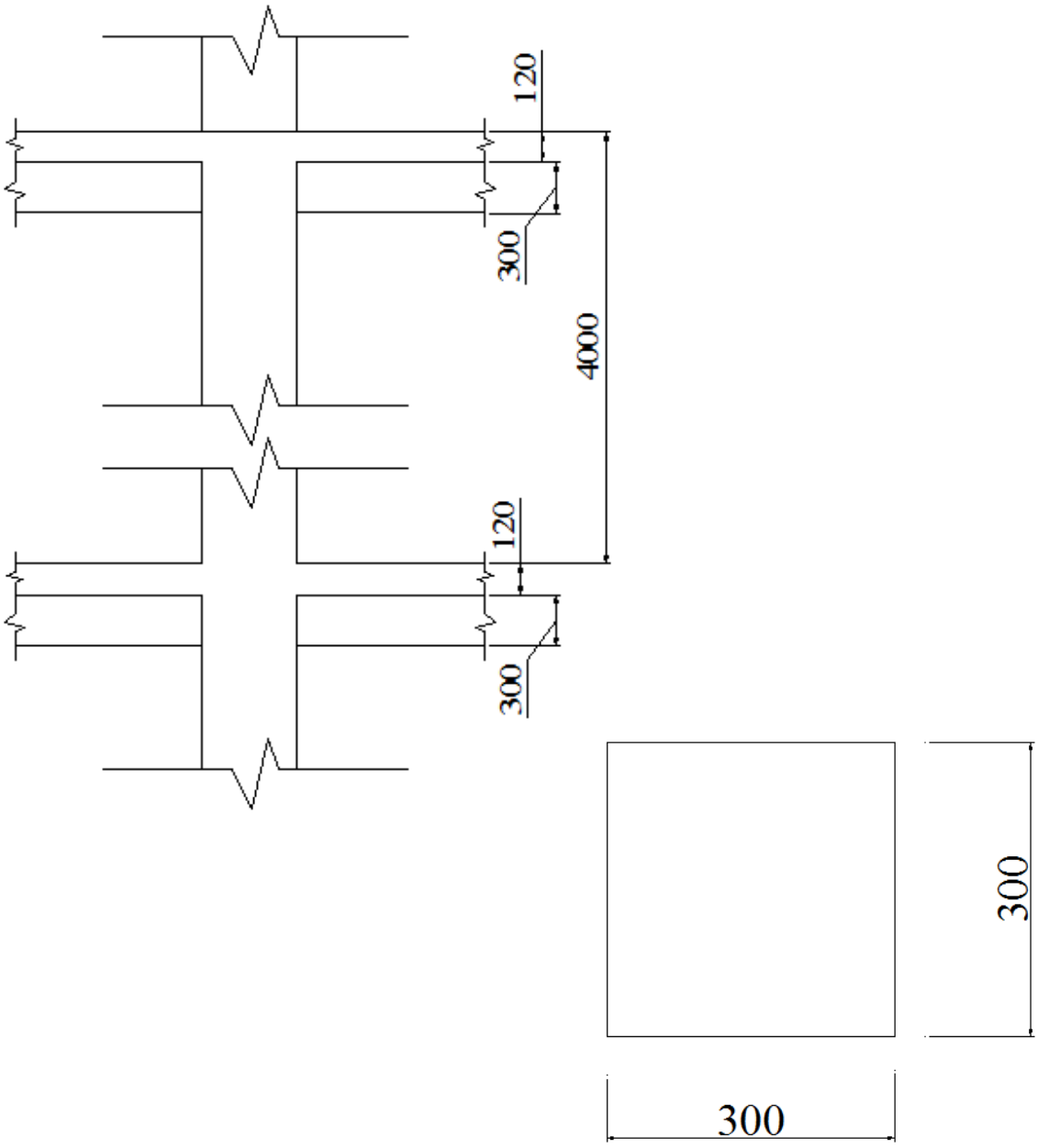


មេរៀនទី៣

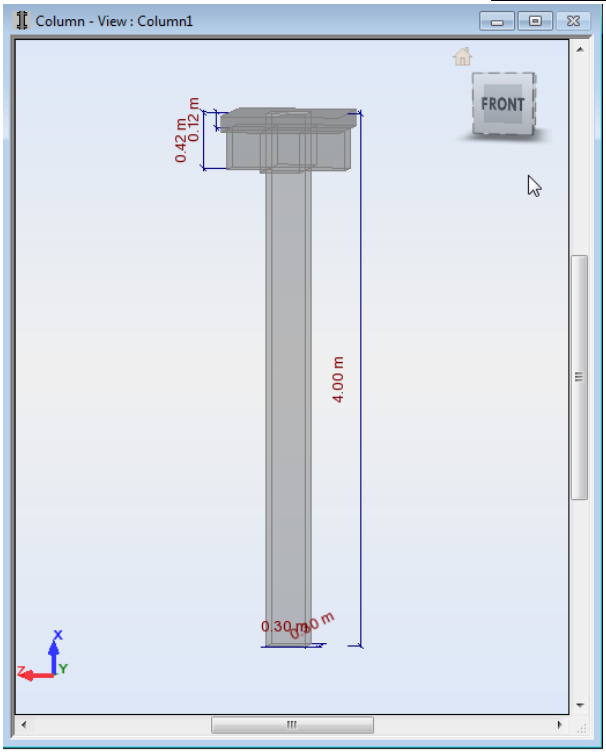
ការគណនាសសរ

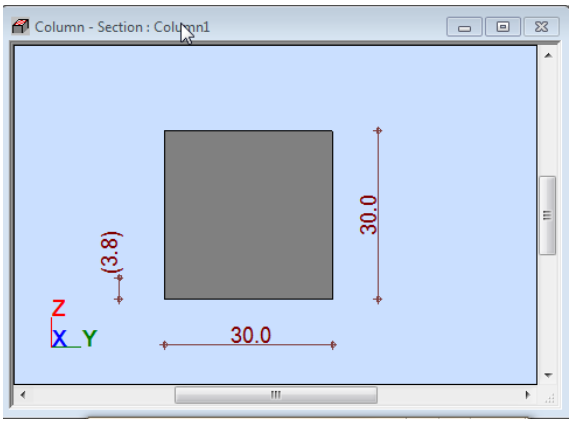
RC Column Design

សសរបេតុងអារមេយកមកគណនា RC Column Calculation :



Plan View





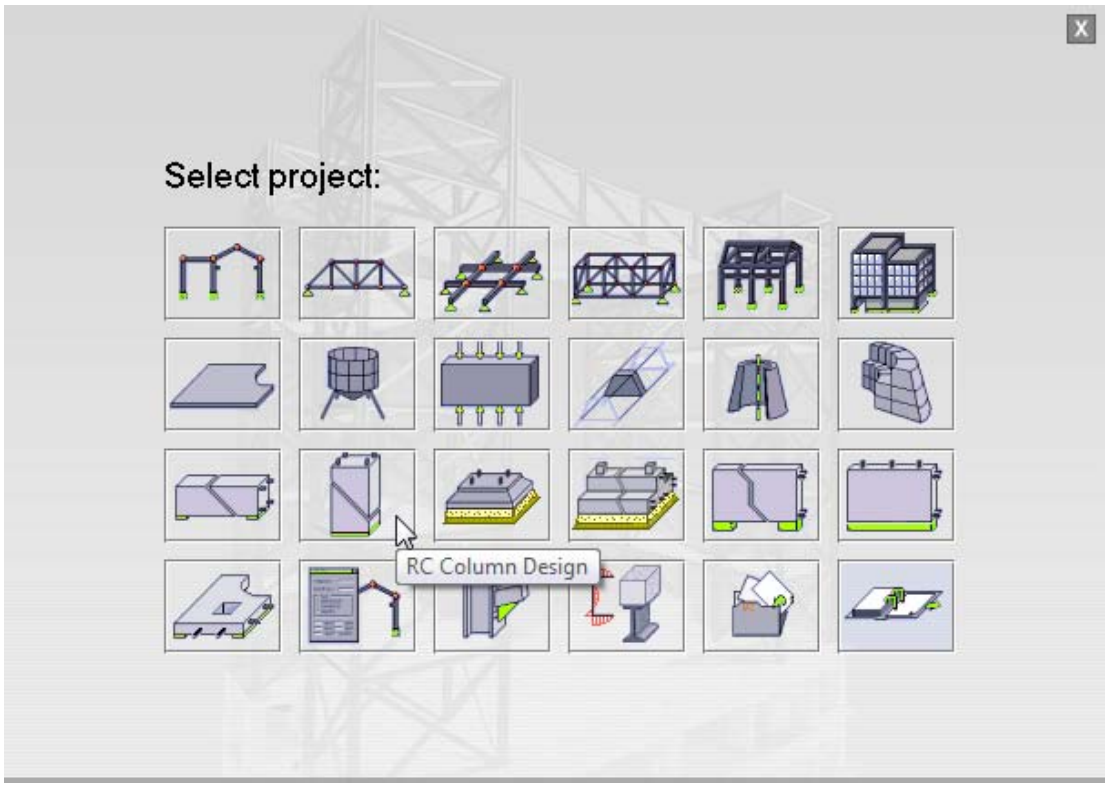
3D View

- **<u>មុខកាត់គ្រឿងបន្ថំ</u>** <u>Frame Section :</u>
  - ការ៉ូ បាយអរ និង ប្លង់សេ : 12cm (Tile + Mortar + Slab)
  - កំពស់ធ្នឹម : 30cm (Beam)
  - មុខកាត់សសរ h = 30cm , h = 30cm (Column Section)
  - កំពស់សសរ L = 4m (Column height)
- **<u>ការកំណត់ប្រភេទបន្ទុក</u>** <u>Determine loading :</u>
  - Dead load អោយតំលៃ : N = 360.0kN , Fy = 0.0 , My = 50.0 , M_Z = 10.0
  - Live load អោយតំលៃ : N = 120.0kN , Fy = 0.0 , My = 25.0 , M_Z = 3.0
- <u>Analysis and Design ដោយ :</u> ACI code
- <u>Material Properties for Analysis and Design :</u>

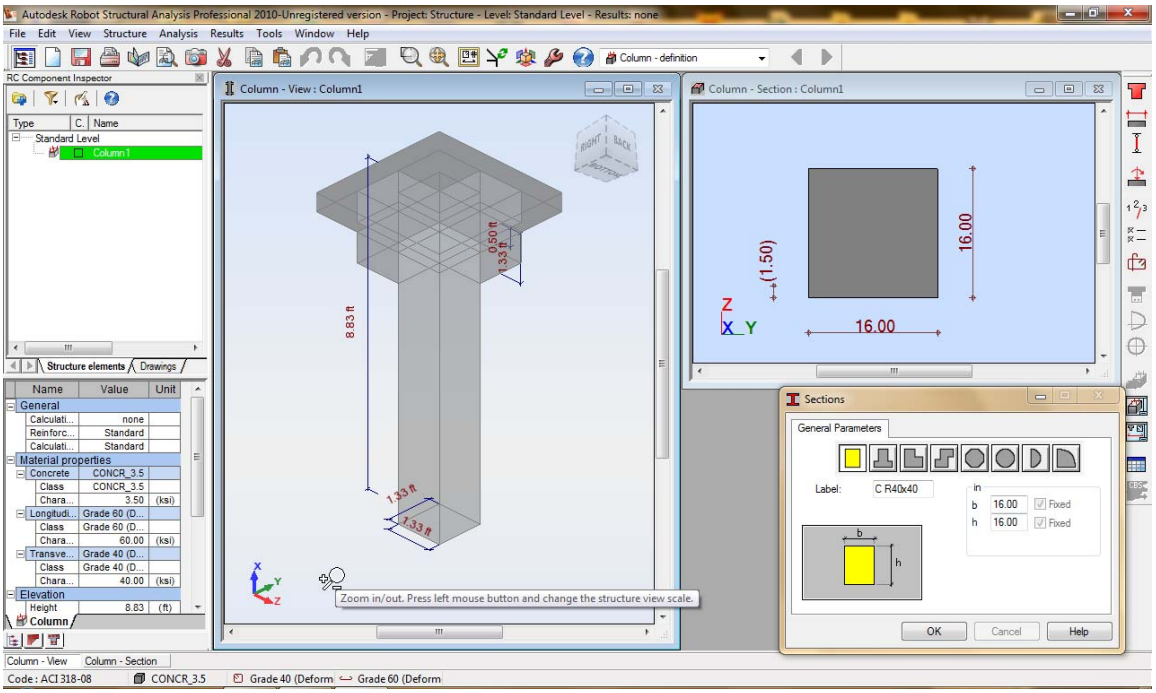
- Yield Strength of Longitudinal Reinforcement : 390 MPa
- Yield Strength of Transversal Reinforcement : 235 MPa
- Compressive Strength of Concrete (Cylinder) : 25 MPa

Step by Step Solution

នៅពេលបើក Program Autodesk Robot Structural Analysis Professional 2010 ដំបូងយើងនិងឃើញដូចរូបខាងក្រោមបន្ទាប់មកយើងជ្រើសរើសយក Icon  ដូចដែលបានបង្ហាញនៅលើរូបខាងក្រោម :

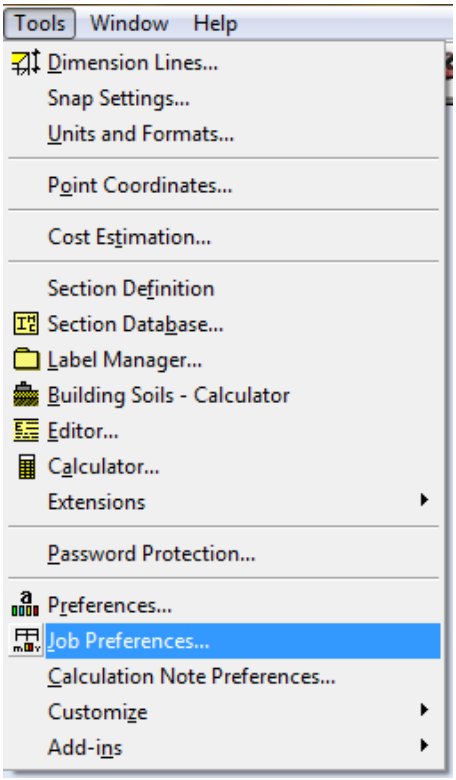


បន្ទាប់មកទៀតអេក្រង់នឹងបង្ហាញដូចរូបខាងក្រោមនេះ

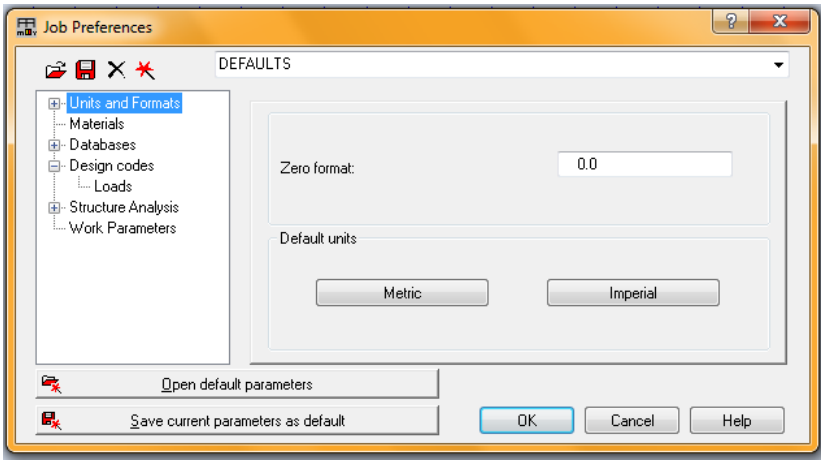


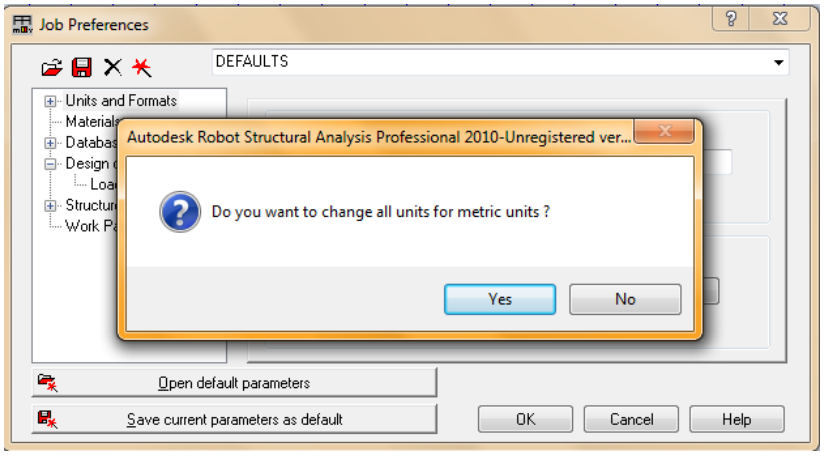
បន្ទាប់មកចូរកំណត់ Units and Format, Materials and Codes មុនពេលធ្វើការ Analysis and Design :

I. ចុចលើ Tool Menu > Job Preference……

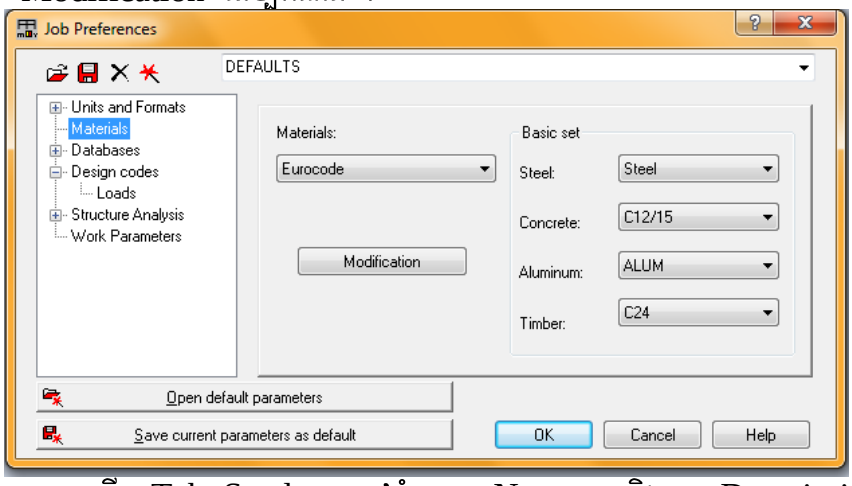


1. អេក្រង់នឹងបង្ហាញផ្ទាំងដូចរូប នៅត្រង់ Units and Formats ត្រូវកំណត់វាជា Metric រួច Yes

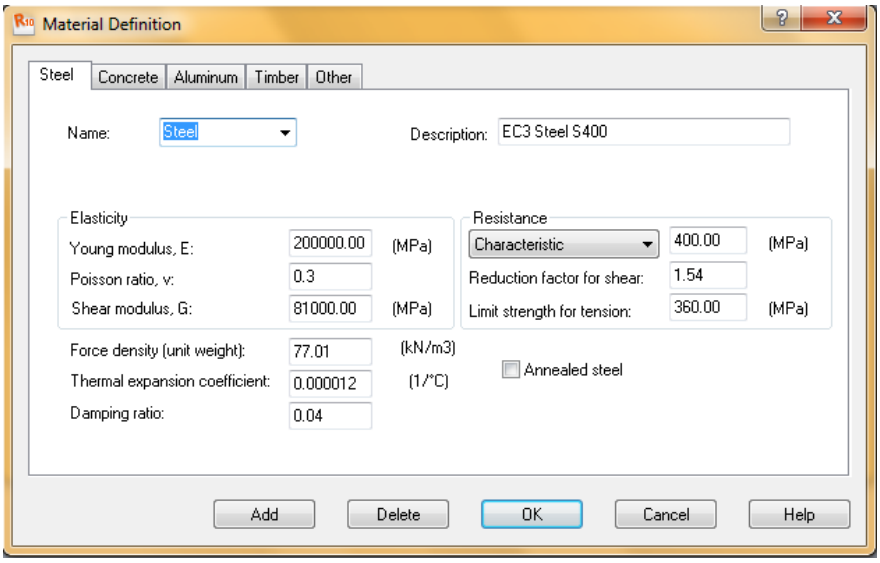




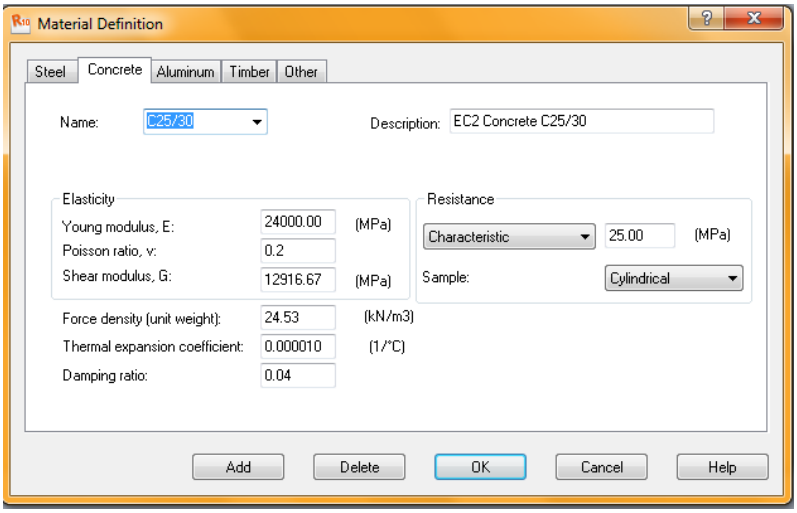
2. នៅត្រង់ Materials ត្រូវកំណត់កូដអោយ Materials ត្រូវយកកូដ Euro code រួចចុចទៅលើ Modification ដើម្បីកំណត់ ។



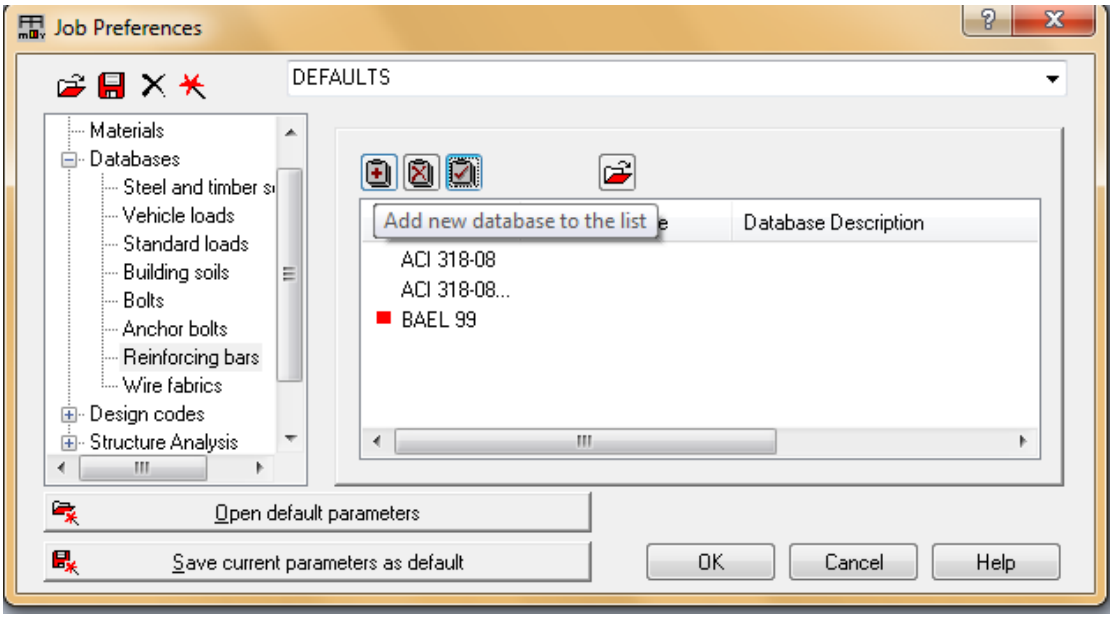
3. រួចចុចលើ Tab Steel រួចត្រូវបំពេញ Name និង Description ដូចខាងក្រោម

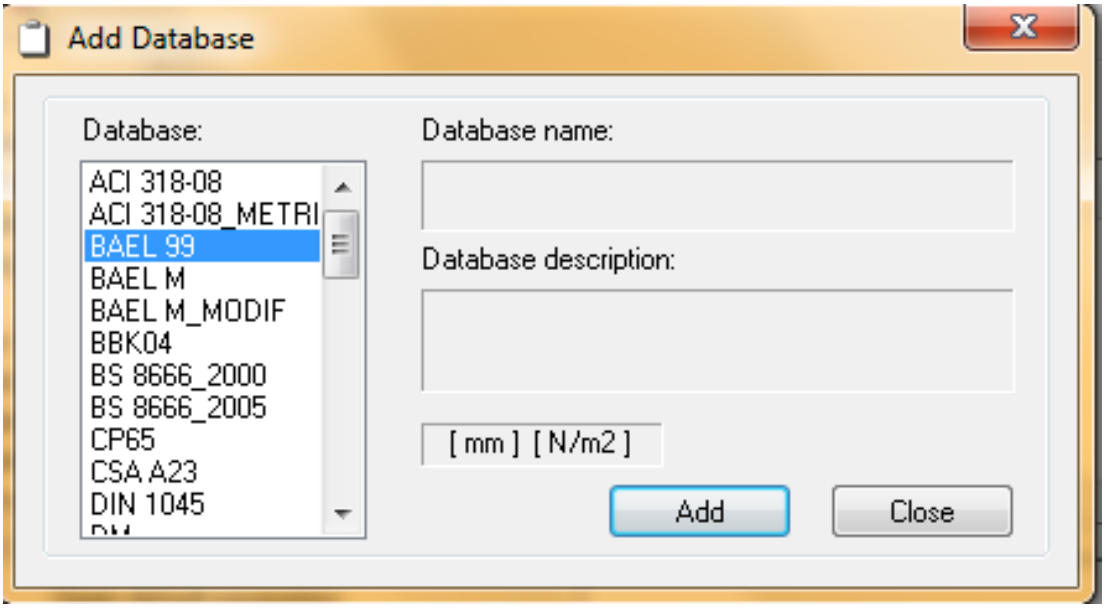


4. រួចចុចលើ Tab Concrete រួចត្រូវបំពេញ Name និង Description និង Young modulus ដូចខាងក្រោម រួចចុច OK ។

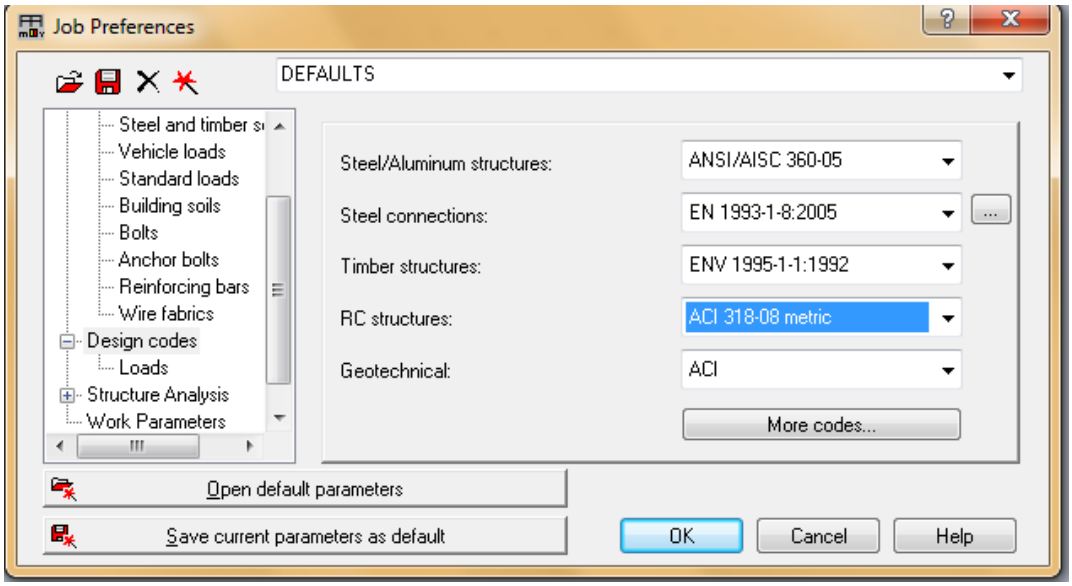


5. រួចចុចលើ Databases រួចរើសយក Reinforcing bars ហើយនៅប្រអប់ខាងស្តាំ ត្រូវបង្កើតកូដ BAEL99 ដោយចុចទៅលើ Icon ដែលមានឈ្មោះថា Add new databases to the list នោះវានឹងចេញតារាង Add Databases រួចរើសយក BAEL99 រួចចុច add ។ បន្ទាប់មករើសកូដដោយយក Mouse select on code រួចចុច Icon ដែលមានឈ្មោះថា Set as the current database ។

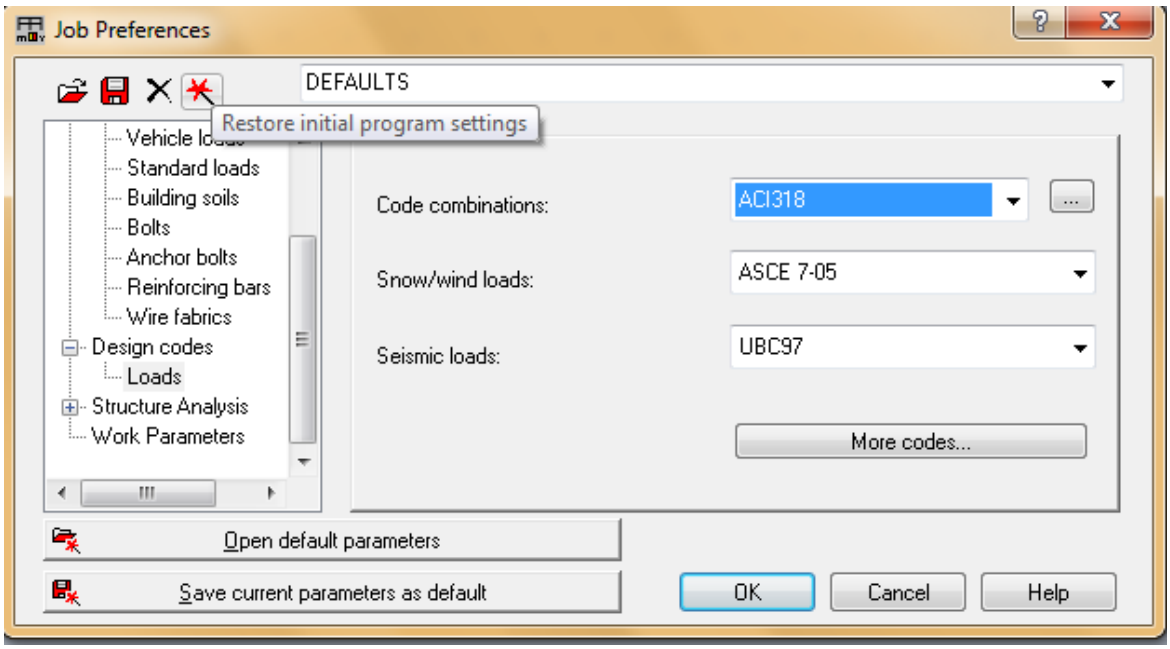




6. រួចចុចលើ Design codes ក្នុងប្រអប់ RC Structures ត្រូវកំណត់ជា ACI 318-08 metric

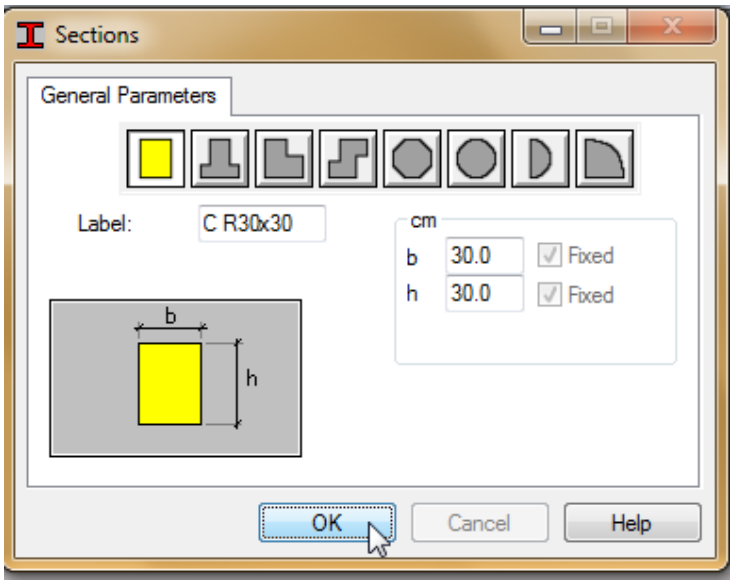


7. នៅត្រង់ Loads ក្នុងប្រអប់ Code combinations ត្រូវកំណត់ជា ACI318 រួចចុច OK ។



I. **ការកំណត់ចុះកាត់សសរ** (Column Section)

ដំបូងត្រូវវាយបញ្ចូលតំលៃមុខកាត់សសរ និងរើសមុខកាត់សសរនៅក្នុងតារាង Section ហើយនៅត្រង់ Label ត្រូវវាយ CR30X30 រួចនៅត្រង់ b និង h ត្រូវវាយតំលៃបញ្ចូល b=30 និង h=30 រួចចុច ok នោះយើងនឹងបានមុខកាត់សសរ ។

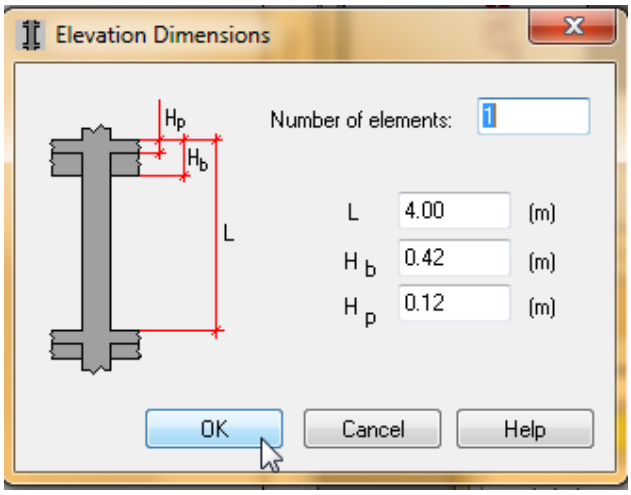


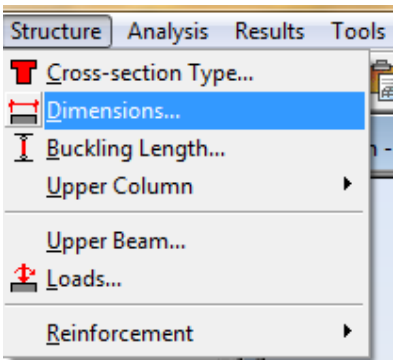
II. **ការកំណត់** Elevation Dimensions :

ដំបូងត្រូវចុចលើ Icon  នៅខាងស្តាំងអេក្រង់ រឺចុចលើ Structure Menu > Dimensions…….



នោះនឹងចេញតារាងដូចខាងក្រោម :





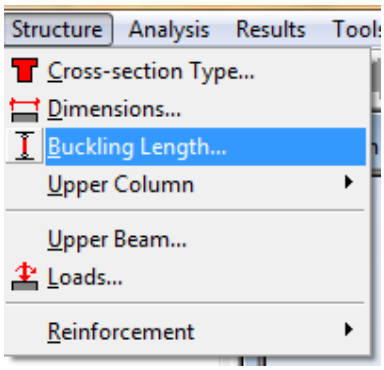
ក្នុងប្រអប់ Elevation Dimension វាយតំលៃ L=4 ជាកំពស់សសរ , Hb=0.42 ជាកំពស់ធ្នឹម និង ប្លង់សេ , Hp=0.12 ជាកំពស់ប្លង់សេ រួចចុច ok ។

III. ការកំណត់ប្រវែងភ្ជាត់ Buckling Length

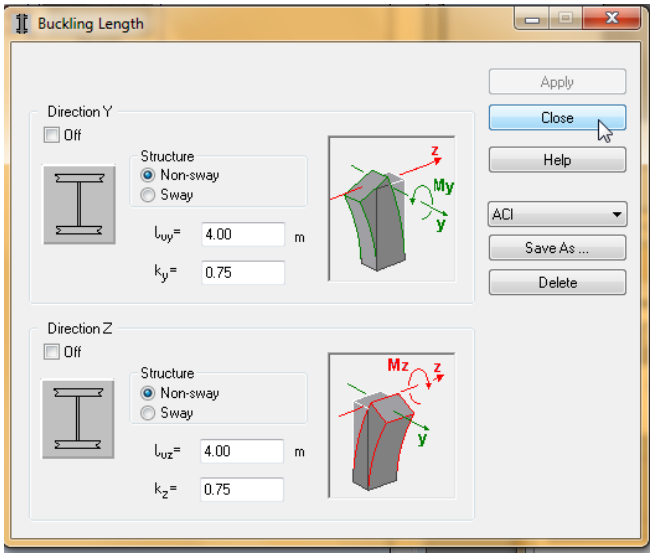
ដំបូងត្រូវចុចលើ Icon  នេះនៅខាងស្តាំងអេក្រង់ រឺចុចលើ Structure > Buckling Length .....

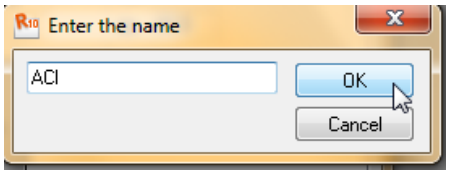


នោះនឹងចេញផ្ទាំងដូចខាងក្រោម : បន្ទាប់មកគ្រាន់តែពិនិត្យមើល



តំលៃមេគុណ k រួចចុច Save As រួចចុច Close ។



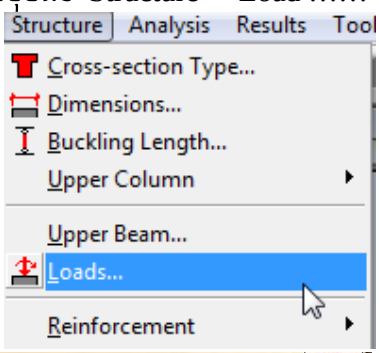


IV. ការកំណត់ប្រភេទបន្ទុក Load Definition

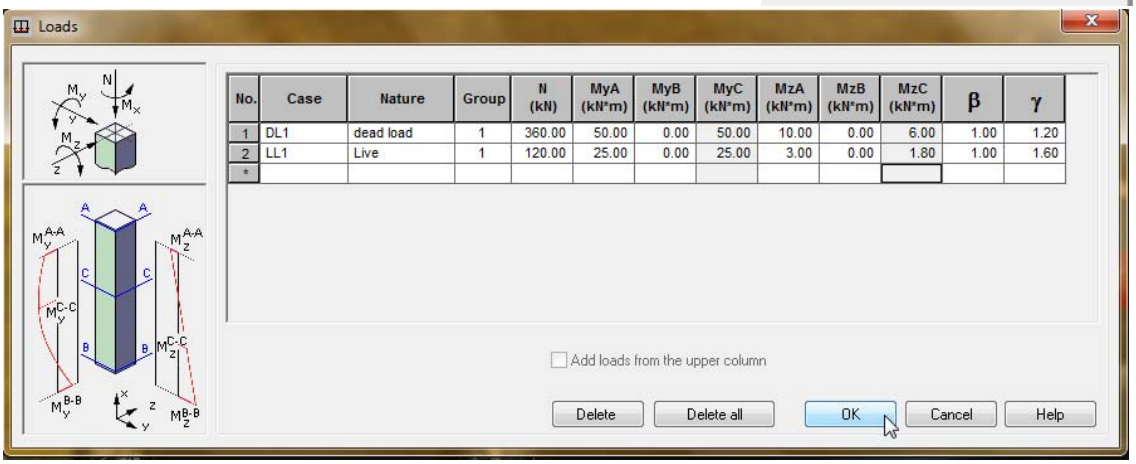
ដំបូងត្រូវចុចលើ Icon  នេះនៅខាងស្តាំងអេក្រង់ រឺចុចលើ Structure > Load .....



នោះនឹងចេញផ្ទាំងដូចខាងក្រោម : បន្ទាប់មកគ្រាន់តែវាយតំលៃបន្ទុក

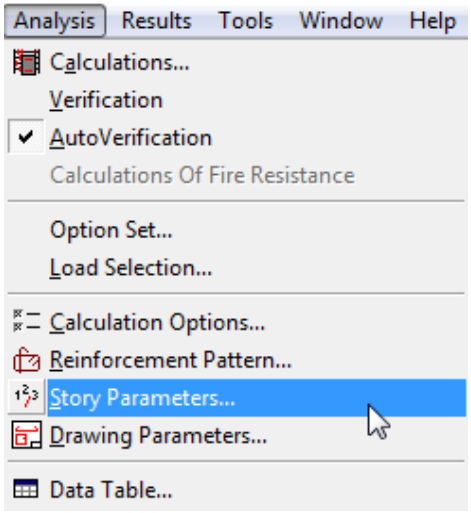


ដែលមាន អំពើលើសសរ រួចចុច ok ។

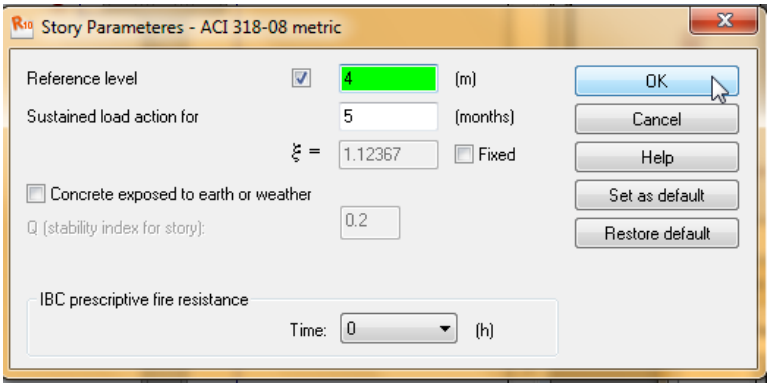


V. ការកំណត់ Story Parameters

ដំបូងត្រូវចុចលើ Icon  នេះនៅខាងស្តាំងអេក្រង់ រឺចុចលើ Analysis > Story Parameters ..... នោះនឹងចេញផ្ទាំងដូចខាងក្រោម : បន្ទាប់មកគ្រាន់តែវាយតំលៃ



Reference Level = 4 រួចចុច ok ។

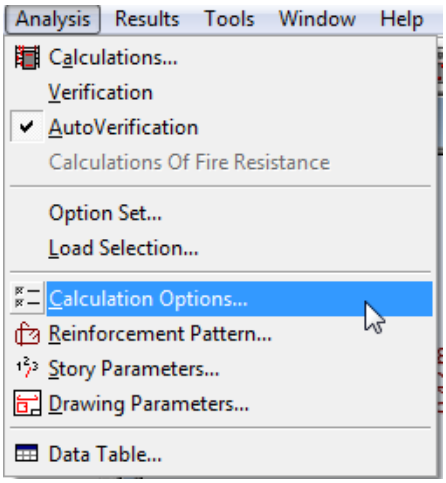


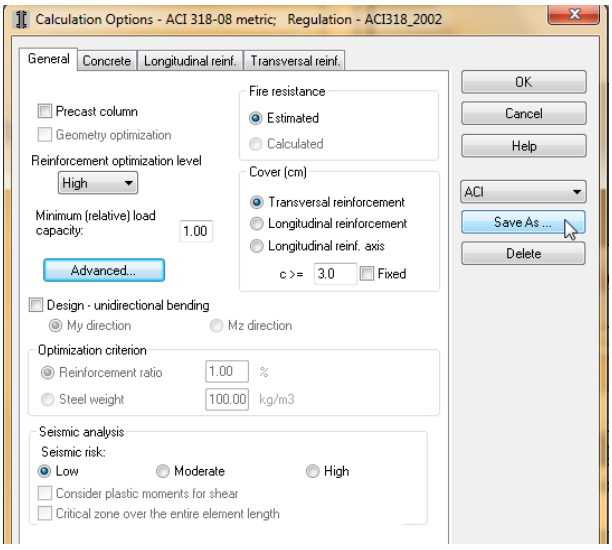
VI. ការកំណត់ Calculation Options

នៅក្នុង Calculation Option មាន ៤ Tab ដែលត្រូវកំណត់

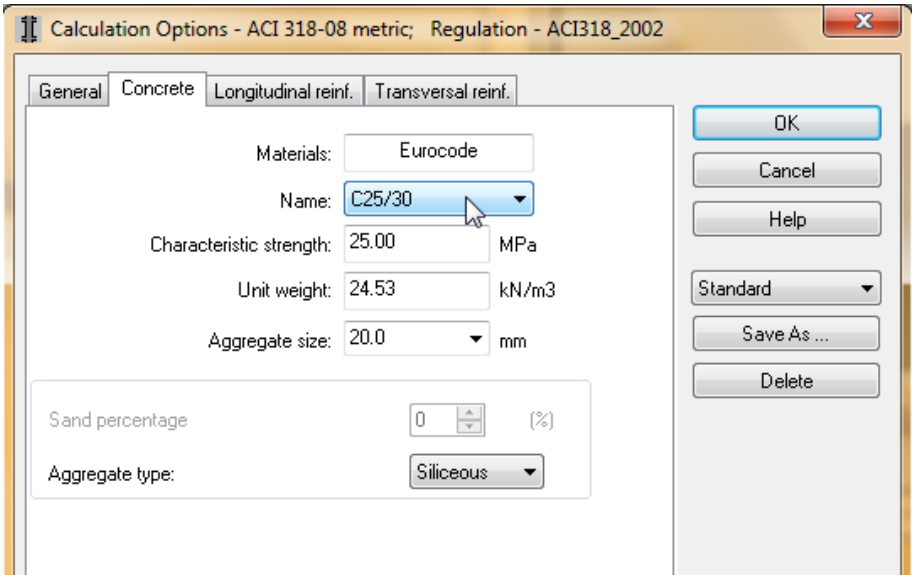
- Tab General : កំណត់ស្រទាប់ការពារ គិតពីប្រវែងសសរចូលគណនា និងការកំណត់ផ្សេងទៀត.....
- Tab Concrete : កំណត់រ៉ែស៊ីស្តង់ សង្កត់របស់បេតុងនៅអាយុ២៨ថ្ងៃ
- Tab Longitudinal reinf : កំណត់រ៉ែស៊ីស្តង់ដែកបណ្តោយ និងអង្កត់ផ្ចិត
- Tab Transversal reinf : កំណត់រ៉ែស៊ីស្តង់ដែកកង និងអង្កត់ផ្ចិត

ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំងអេក្រង់ រឺចុចលើ Analysis Menu > Calculation Option ..... នោះនឹងចេញផ្ទាំងដូចខាងក្រោម : Tab General

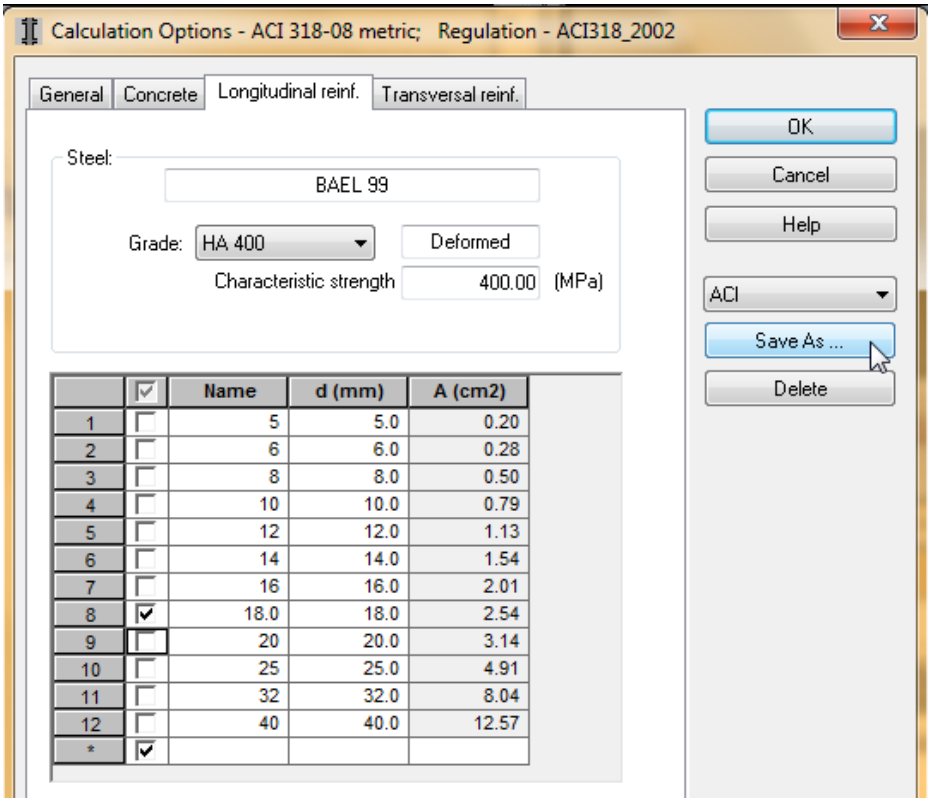




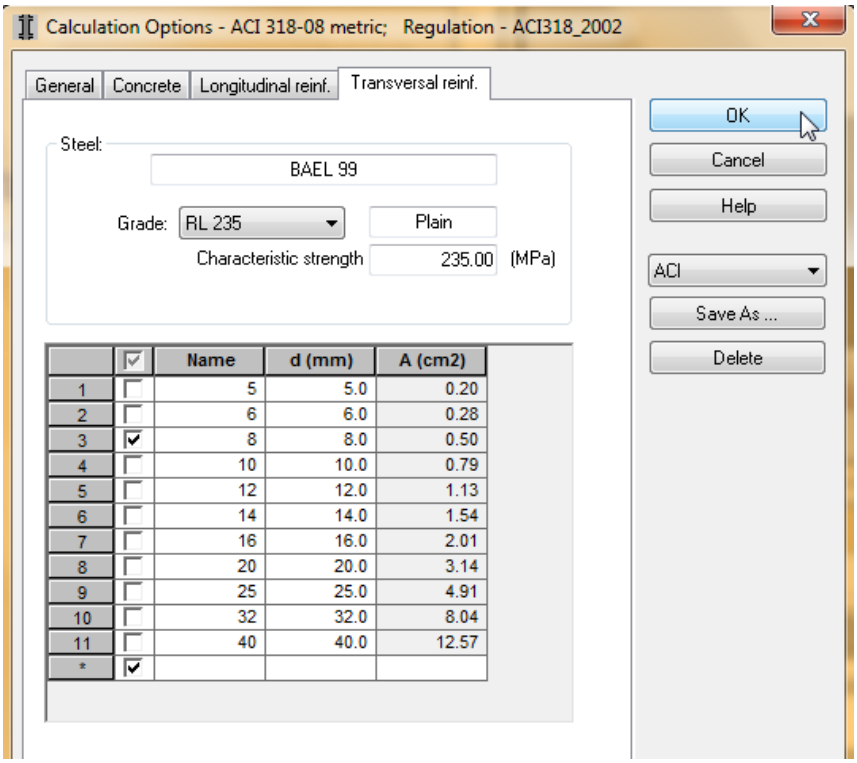
Tab Concrete :



Tab Longitudinal reinf…..



Tab Transversal reinf…..



នៅពេលកំណត់រួចរាល់ហើយត្រូវចុចលើ Save As វាឈ្មោះ ACI រួចចុច ok ដើម្បីរក្សាទុក ។

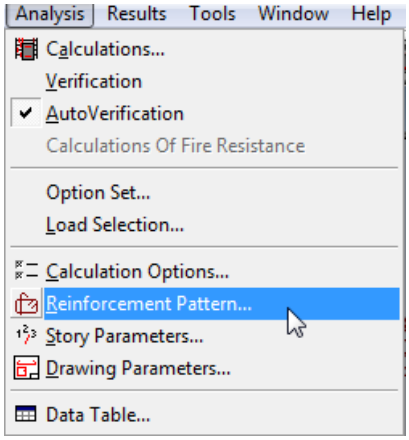
VII. **ការកំណត់** Reinforcement Pattern

នៅក្នុង Reinforcement Pattern មាន ៤ Tab ដែលត្រូវកំណត់

- Tab Longitudinal bars : កំណត់គំលាតពីសសៃដែក១ទៅសសៃដែក១ ការចងដែក……
- Tab Transversal bars : កំណត់គំលាតចងដែកកង និងការពត់ដែកកង ……
- Tab Dowel bars : កំណត់ប្រវែងដែកសំរាប់ទុកអោយតភ្ជាប់ទៅសសរខាងលើ……
- Tab Shapes : កំណត់ទ្រង់ទ្រាយ លក្ខណៈនៃការចងដែក……

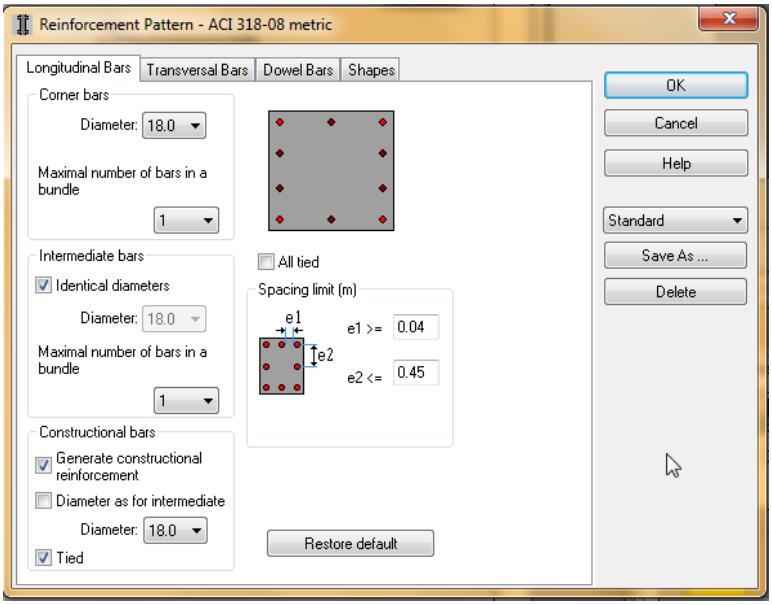
ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំអេក្រង់ រីចុចលើ Analysis Menu > Reinforcement Options …….

នោះនឹងចេញផ្ទាំងដូចខាងក្រោម :

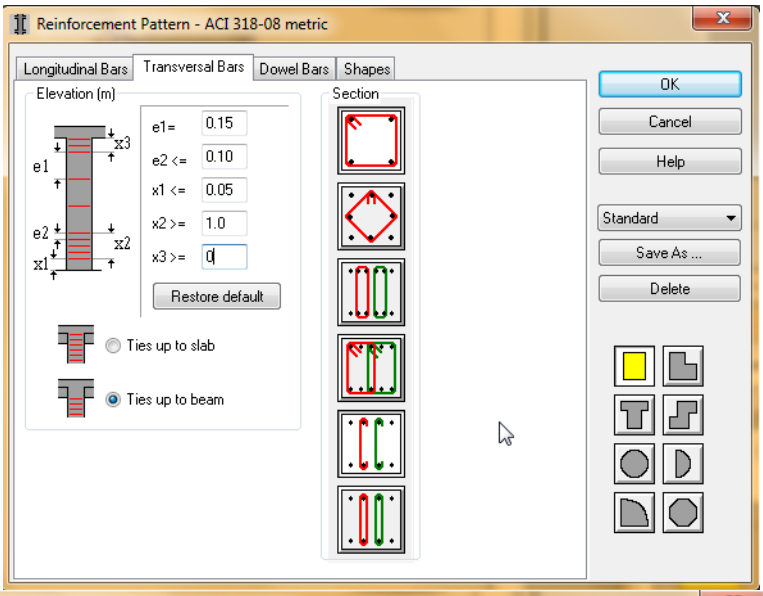




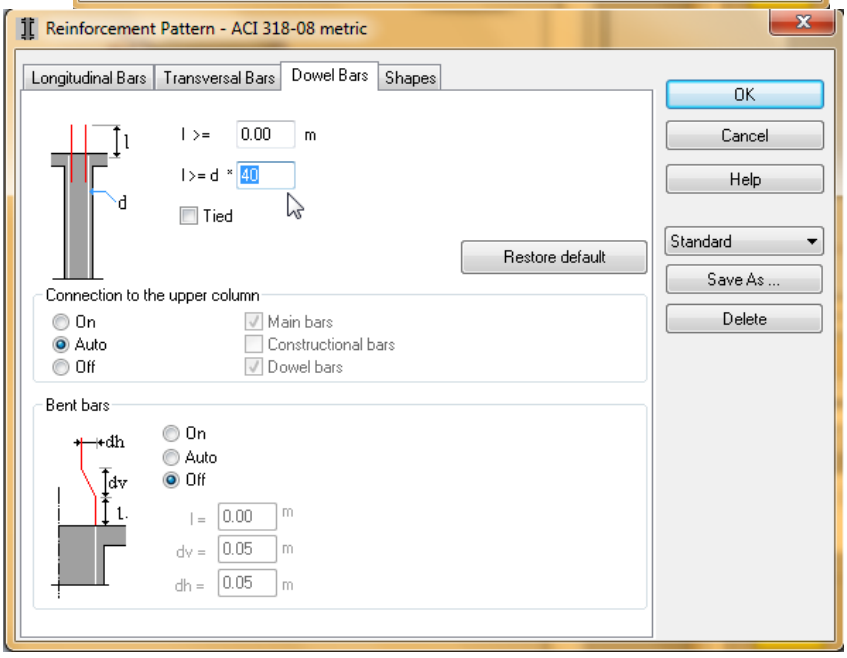
Tab Longitudinal bars:



Tab Transversal bars :

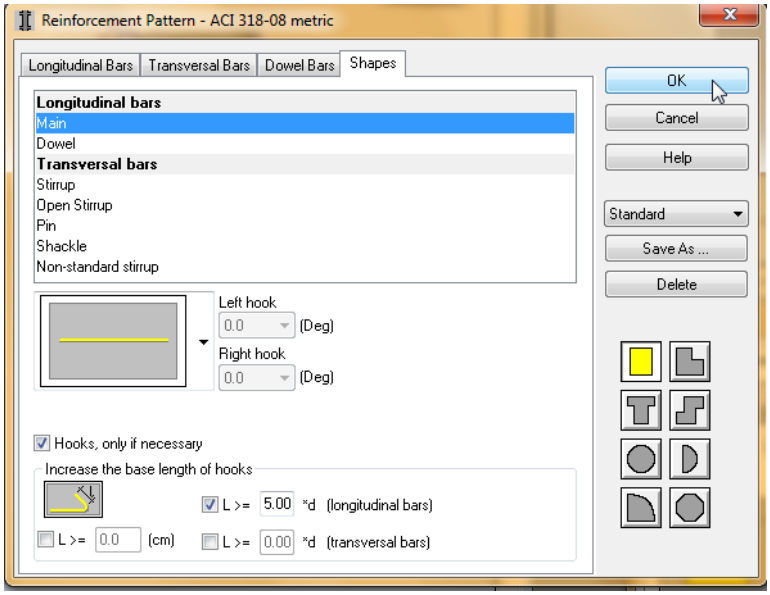


Tab Dowel bars :



www.yerngshare.blogspot.com

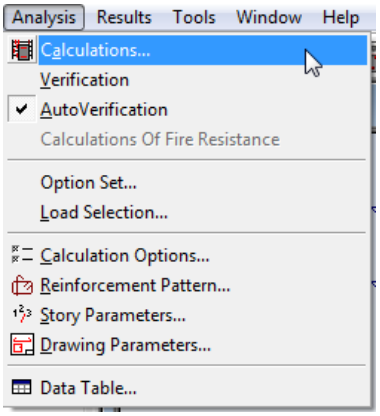
Tab Shapes :



នៅពេលកំណត់រួចរាល់ហើយត្រូវចុចលើ Save As វាឈ្មោះ ACI រួចចុច ok ដើម្បីរក្សាទុក ។

VIII. **ចាប់ផ្តើមគណនារកសសៃថែក** Start Calculate

ដំបូងចុចលើ Icon  នេះនៅខាងលើអេក្រង់ រឺចុចលើ Analysis Menu > Calculation ……..

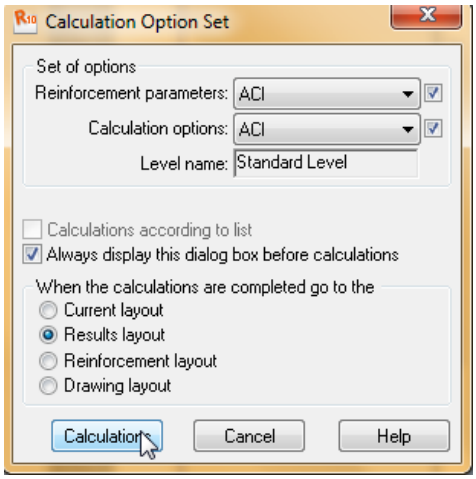


នោះនឹងចេញផ្ទាំងដូចខាងក្រោម : បន្ទាប់មកនៅត្រង់កន្លែង

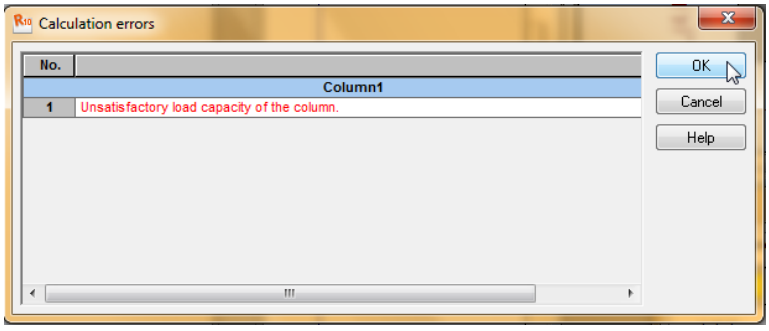
Reinforcement parameters : ជ្រើសរើសយកឈ្មោះដែលយើងបាន Save ទុក (ACI)

Calculation options : ជ្រើសរើសយកឈ្មោះដែលយើងបាន Save ទុក (ACI) រួចចុច

Results Layout ដើម្បីបង្ហាញលទ្ធផល រួចចុច Calculation ។



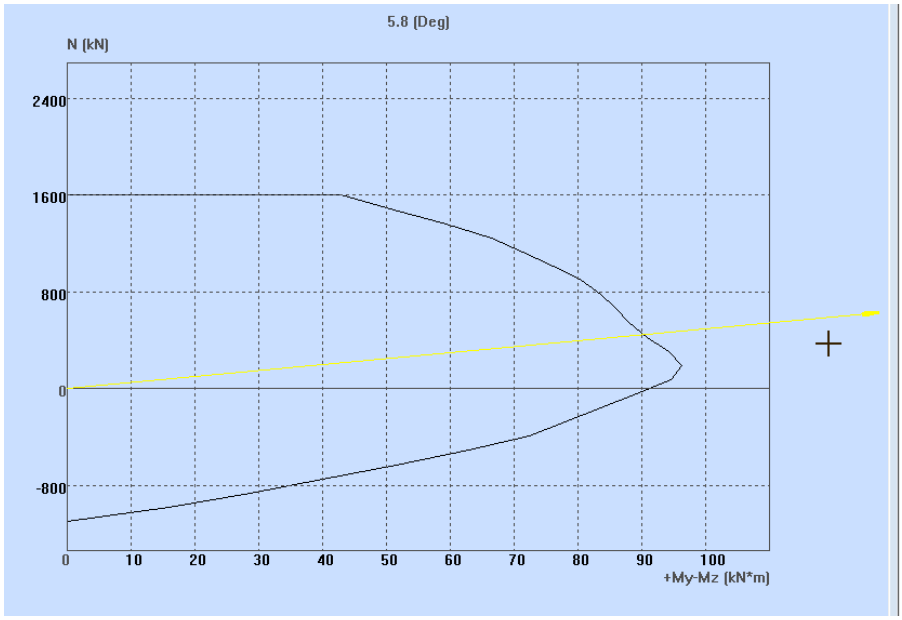
បន្ទាប់មកវានឹងបង្ហាញផ្ទាំងដូចខាងក្រោម :

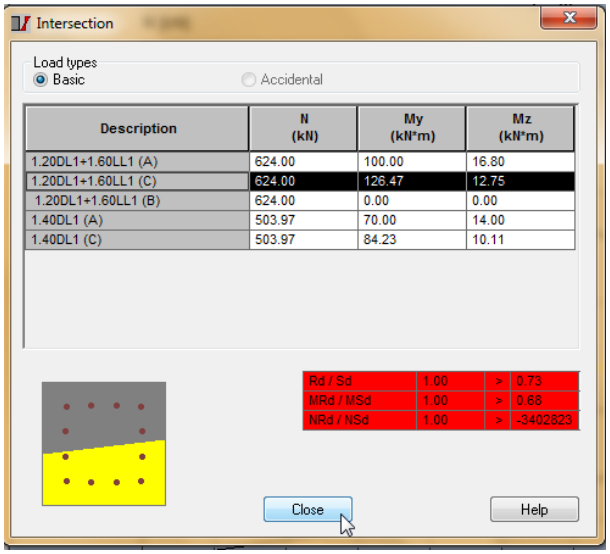


ផ្ទាំងនេះបញ្ជាក់ថា ការគណនារបស់យើងគឺមានការ errors កន្លែងណាមួយ ។

បន្ទាប់មកចុច ok វានឹងចេញដ្យាក្រាមបង្ហាញពីការ errors របស់សសរ ដោយមើលទៅលើសញ្ញាព្រួញពណ៌លឿងចេញពីរង្វង់ដ្យាក្រាម ។ បើសិនជា សញ្ញាព្រួញពណ៌លឿងស្ថិតនៅក្នុងរង្វង់ដ្យាក្រាមវិញនោះគឺ បានន័យថាសសរយើងគ្មានការ errors ទេ ។

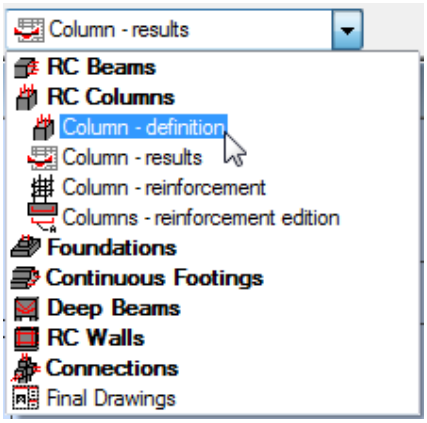
www.yermgshare.blogspot.com





តំលៃនៅក្នុងតារាងទាំងនេះតួចជាងសូន្យបានន័យថាសសរយើង errors មិនជាប់ ឬ មិនជាប់ ។

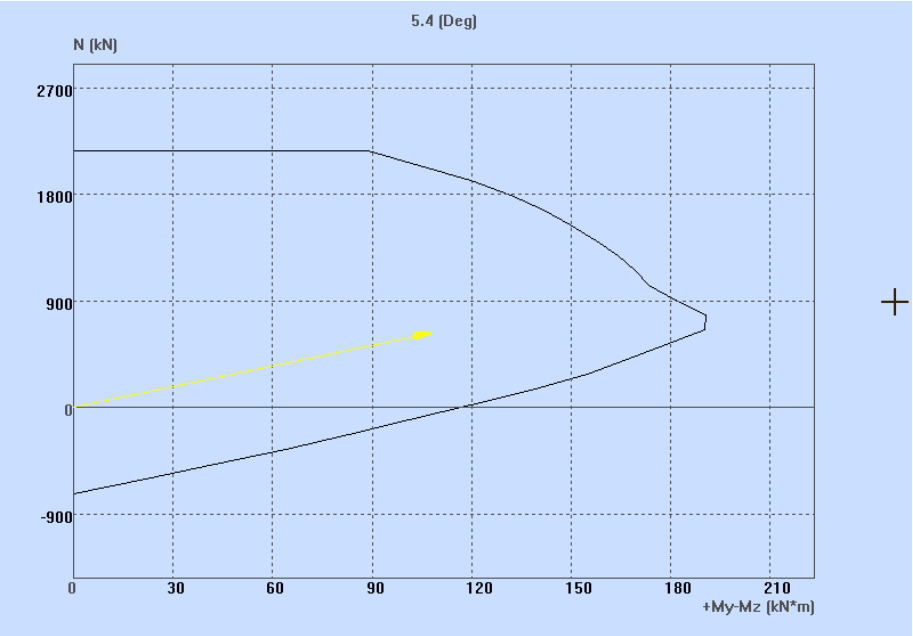
បើសិនជា errors ដូចខាងលើយើងត្រូវត្រលប់ចូលទៅកែសសៃដែក រឺក៏កែមុខកាត់សសរនៅក្នុង Column –definition នៅក្នុង Layout ខាងលើដូចរូប ។

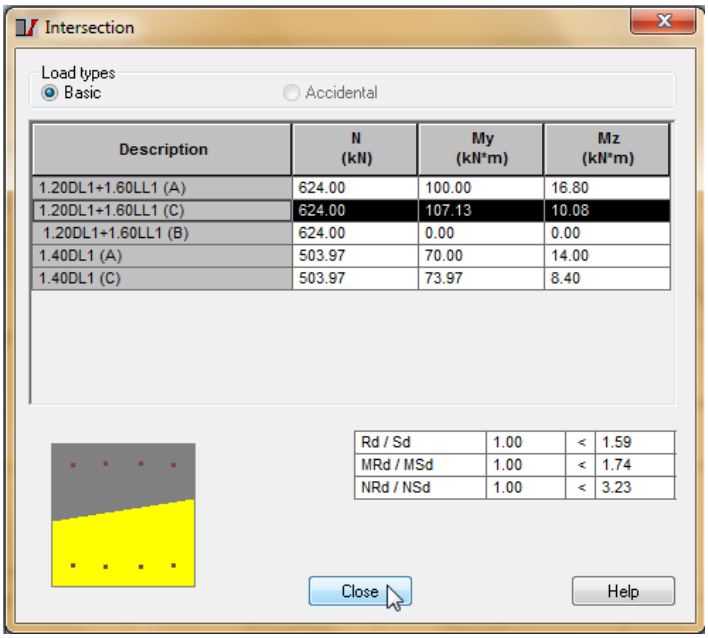


បន្ទាប់មកចូលទៅកែមុខកាត់ដែកនៅក្នុង Calculation Options ដោយចុចទៅលើ Icon នេះ ។ រួចធ្វើការគណនាដូចមុនម្តងទៀត ដោយចុចទៅលើ Icon នេះ ។

បន្ទាប់ពីកែមុខកាត់សសរពី b = 30 និង h = 30 ឱ្យទៅជា b = 40 និង h = 40 រួចយើងធ្វើការគណនាឡើងវិញដោយចុចទៅលើ Icon នេះ ។ បន្ទាប់មកយើងឃើញសញ្ញាព្រួញពណ៌លឿងនៅក្នុងរង្វង់បានន័យថាសសរយើងមិន errors ទេគឺវាអាចទប់នឹងបន្ទុកដែលមានអំពើមកលើវាបាន ។

www.yerngshare.blogspot.com

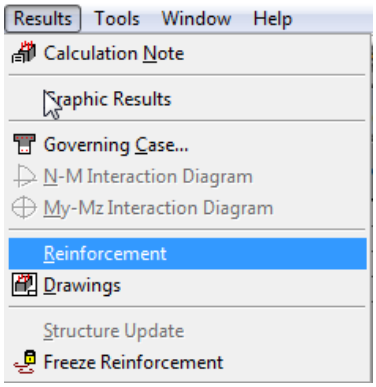


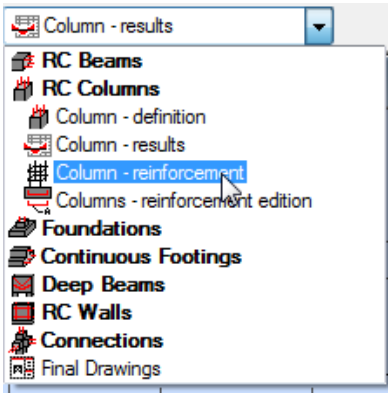


តំលៃនៅក្នុងតារាងទាំងនេះធំជាងសូន្យបានន័យថា សសរយើងមិន errors ទេគឺជាប់ អាចទប់ទល់កំលាំងដែលមានអំពើមកលើវា។

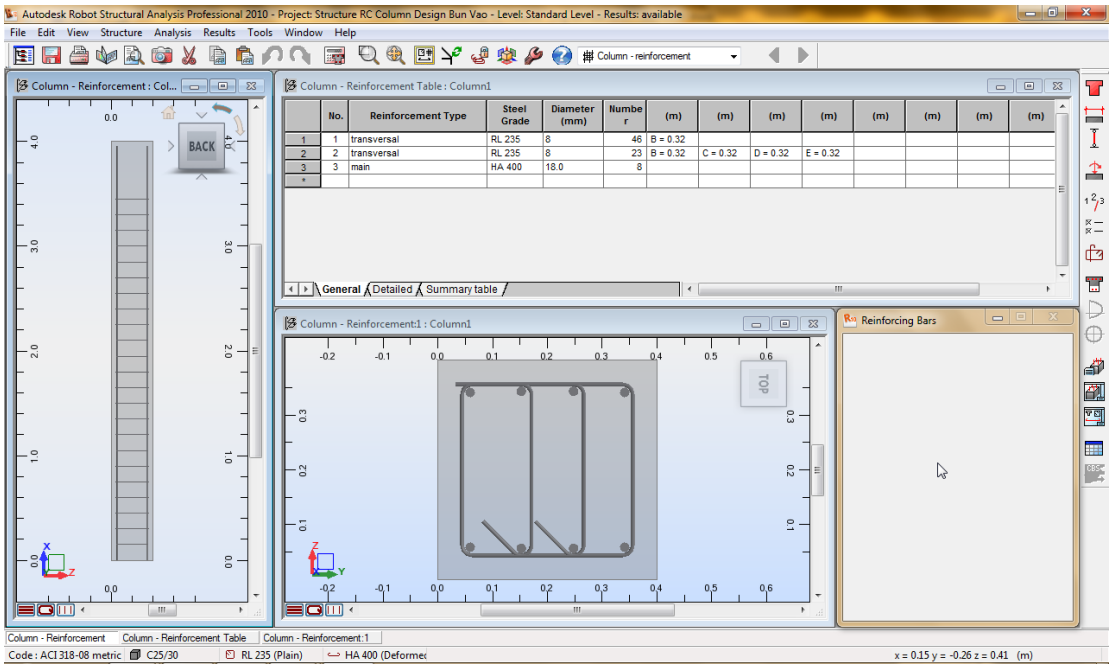
IX. ការត្រួតពិនិត្យមើលការរៀបចំសៀវែក Column - Reinforcement

ដំបូងចុចលើ Result Menu > Reinforcement រឺចុចលើ Layout ដូចរូប



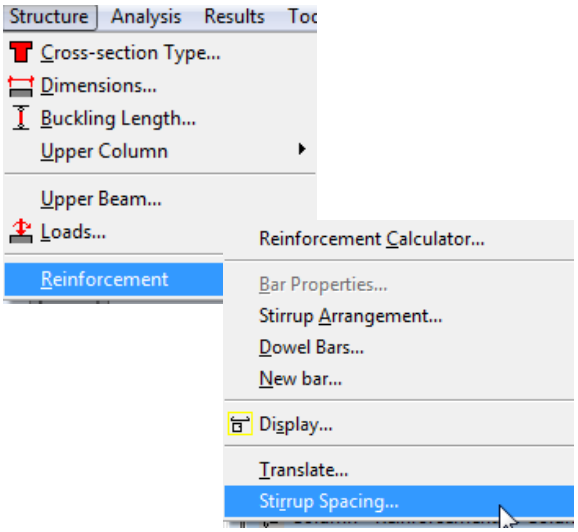


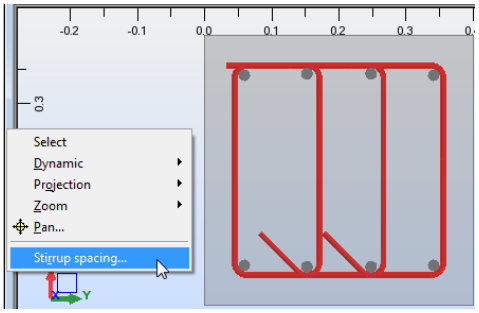
នោះនឹងបង្ហាញដូចរូបខាងក្រោម :

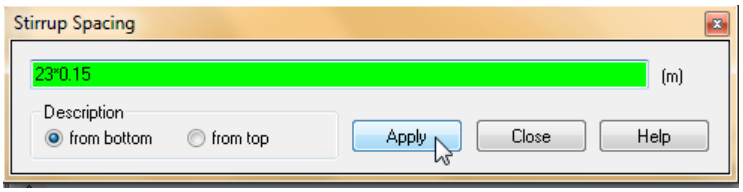


ការកែប្រលោះរបស់ដែកកង Stirrup Spacing :

ដំបូង Select ដែកកងទាំងអស់ដែលមានឱ្យចេញពណ៌ក្រហម រួចចុច Right Click លើអេក្រង់រើសយក Stirrup Spacing រឺក៏ចុចលើ Structure > Reinforcement > Stirrup Spacing ដូចរូប ខាងក្រោម

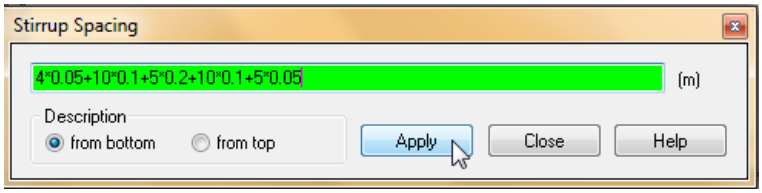




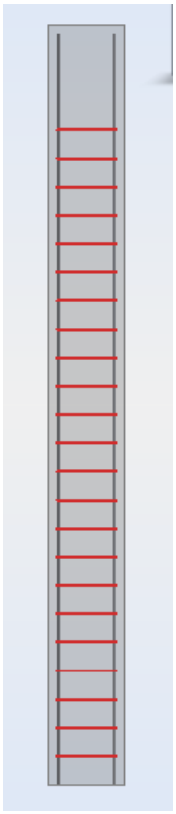
នោះនឹងចេញផ្ទាំងដូចនេះ 

នៅពេលមិនទាន់កែ

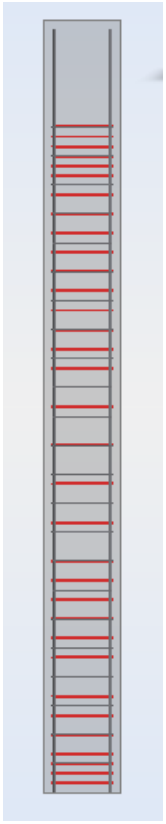
រួចសរសេរលេខ ៖ 4*0.05+10*0.1+5*0.2+10*0.1+5*0.05 បន្ទាប់មកចុច Apply



ក្រោយពេលកែរួច



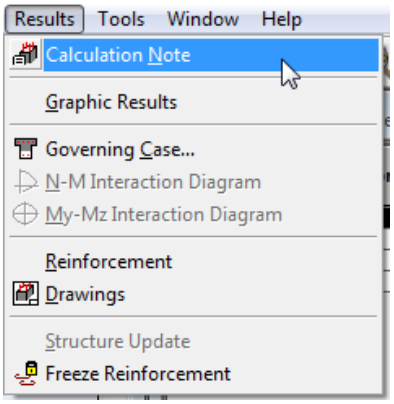
➢ មុនពេលកែ



> ក្រោយពេលកែរួច

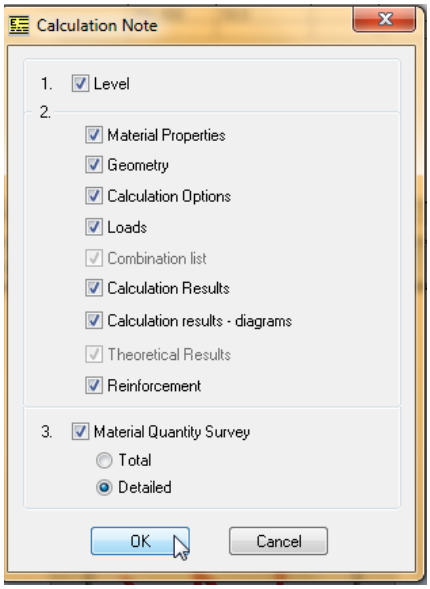
X. Calculation Note

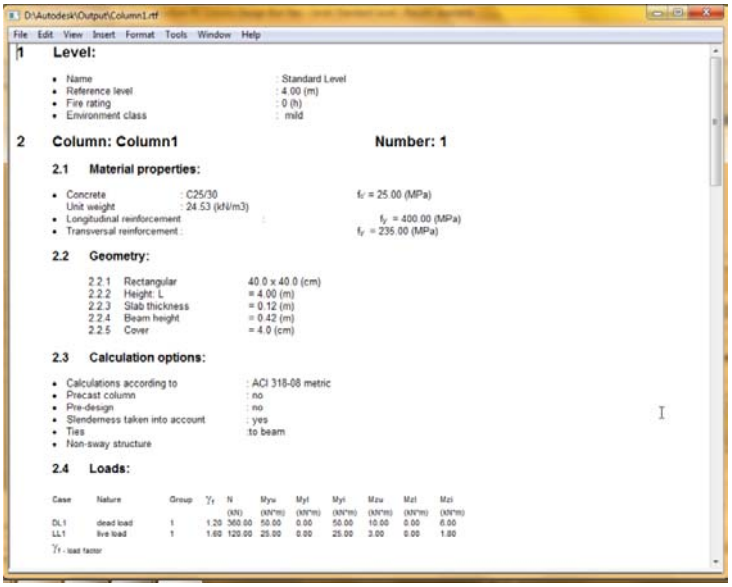
ដំបូងចុចលើ Icon  នេះនៅស្តាំអេក្រង់រីចុចលើ Analysis Menu > Calculation Note





បន្ទាប់មកនឹងចេញតារាងនេះ : ក្រោយពីចុច ok នៅក្នុងតារាង Calculation Note នឹងចេញតារាងខាងក្រោមនេះសរាប់បញ្ជាក់ពតិមានអំពីសសរដែលយើងបានធ្វើ ។



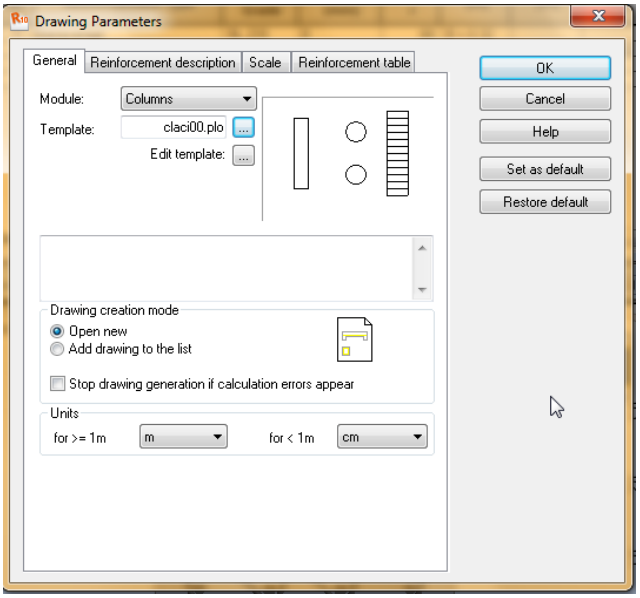


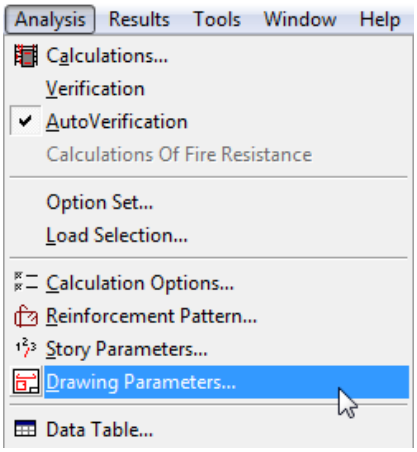
XII Drawing Parameters **ការរៀបចំក្រចាស់មើលឆ្លង់**

ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំអេក្រង់រីចុចលើ Analysis > Drawing Parameters

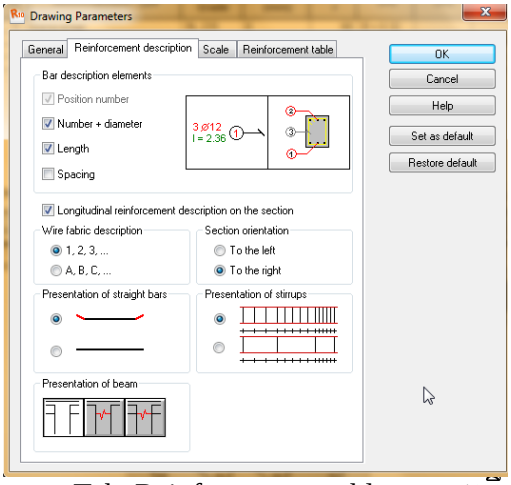


Tab General :

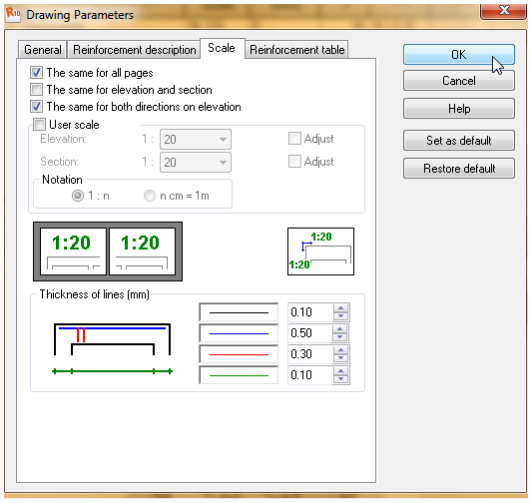




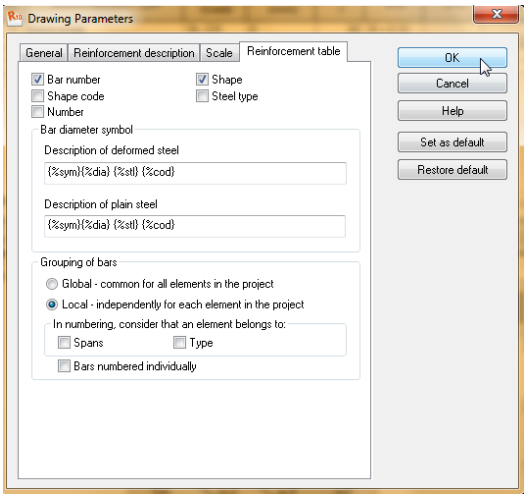
Tab Reinforcement description : ការជ្រើសរើសមើល ប្រវែង គំលាត មុខកាត់ដែក ។



Tab Scale : ការពង្រីកមើលតំលៃលេខ

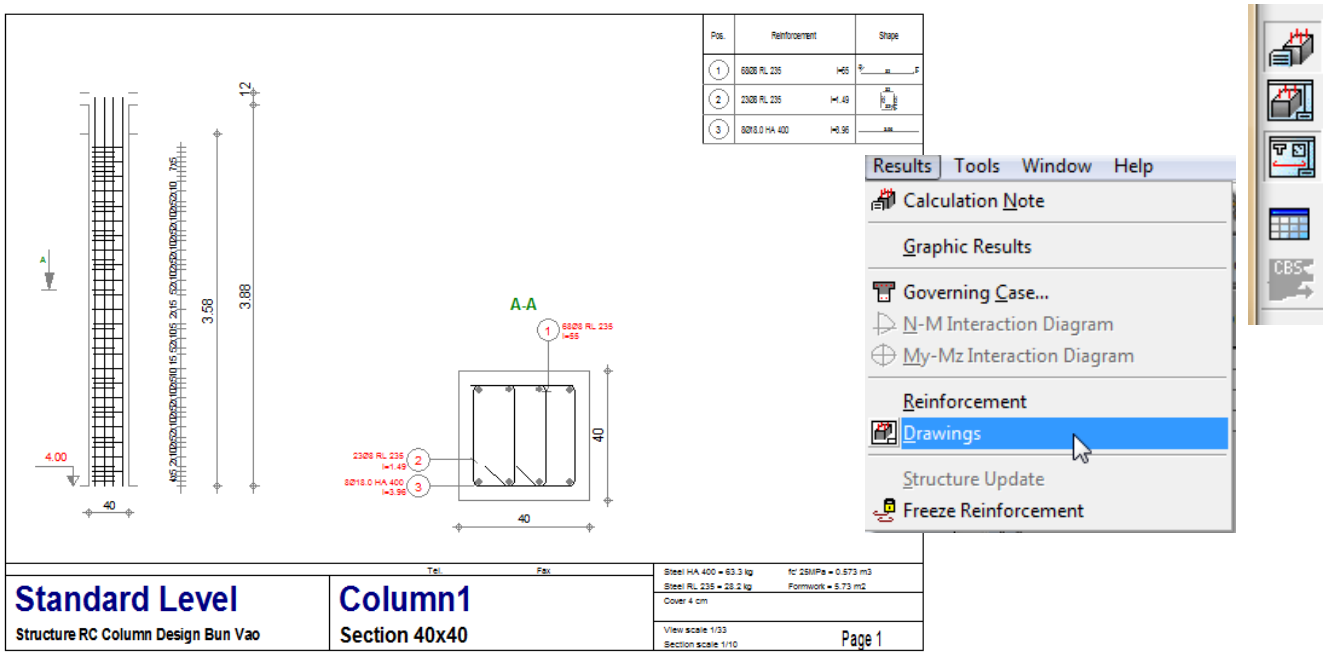


Tab Reinforcement table : ការមើលទ្រង់ទ្រាយដែក



XIII Column Drawings **ការមើលសសរ**

ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំអេក្រង់រីចុចលើ Result Menu > Drawings ដូចរូប :

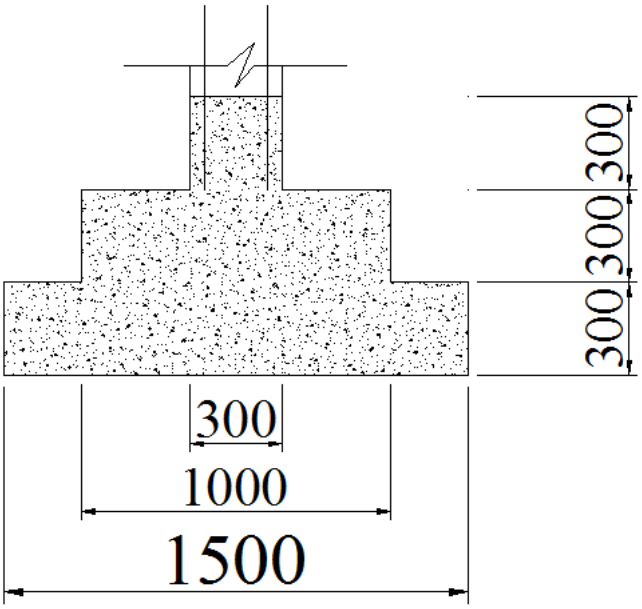


មេរៀនទី៤

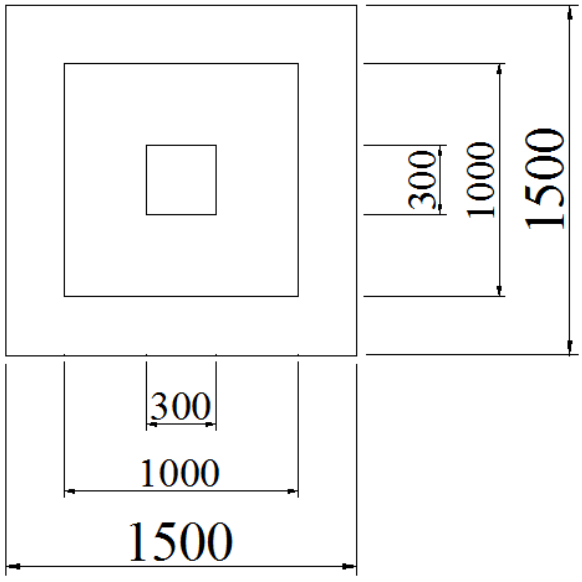
ការគណនាគ្រឹះរាក់

Shallow Foundation Design

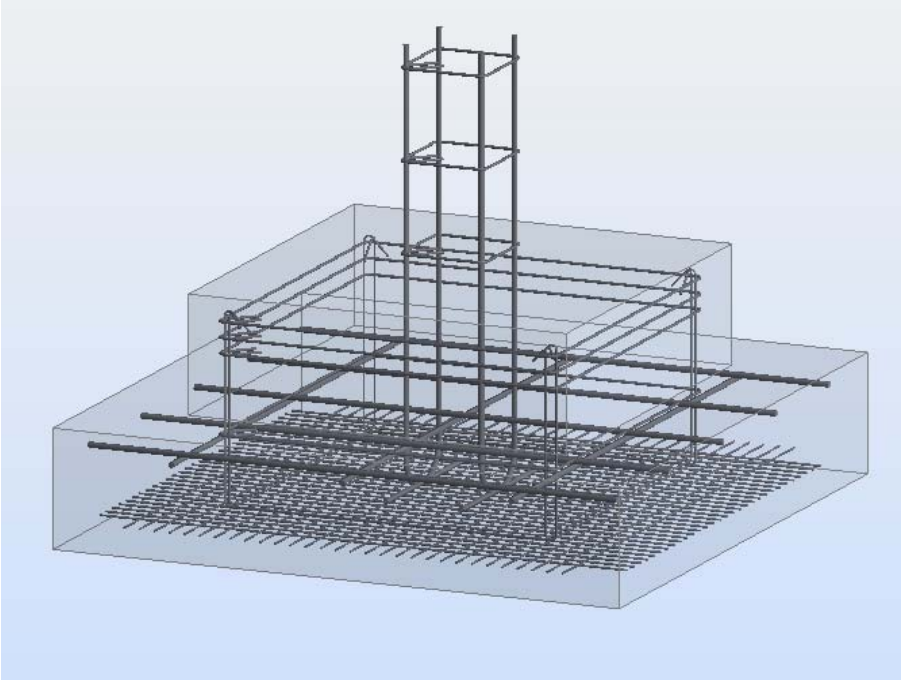
គ្រឹះរាក់ដែលយកមកគណនា Shallow Foundation Calculation :



Elevation View



Plan View



3D View

- **<u>មុខកាត់គ្រឿងបង្គុំ</u>** Frame Section :
  - គណនាលទ្ធភាពទ្រទ្រង់របស់គ្រឹះរាក់ដែលមាន : ទំហំគ្រឹះធ្វើជាពីរថ្នាក់ 1.5m x 1.5m និង 1m x 1m កំរាស់របស់ថ្នាក់នីមួយៗ 0.3m ស្ថិតក្នុងស្រទាប់ដី Silty Clays មានកំរាស់ 0.6m និង Sandy Clay មានកំរាស់ 2m , Clayey Sandy មិនកំណត់កំរាស់ ដែលបាតរបស់គ្រឹះស្ថិតនៅក្នុងជំរៅ 2m ។
- **<u>ការកំណត់ប្រភេទបន្ទុក</u>** Determine loading :

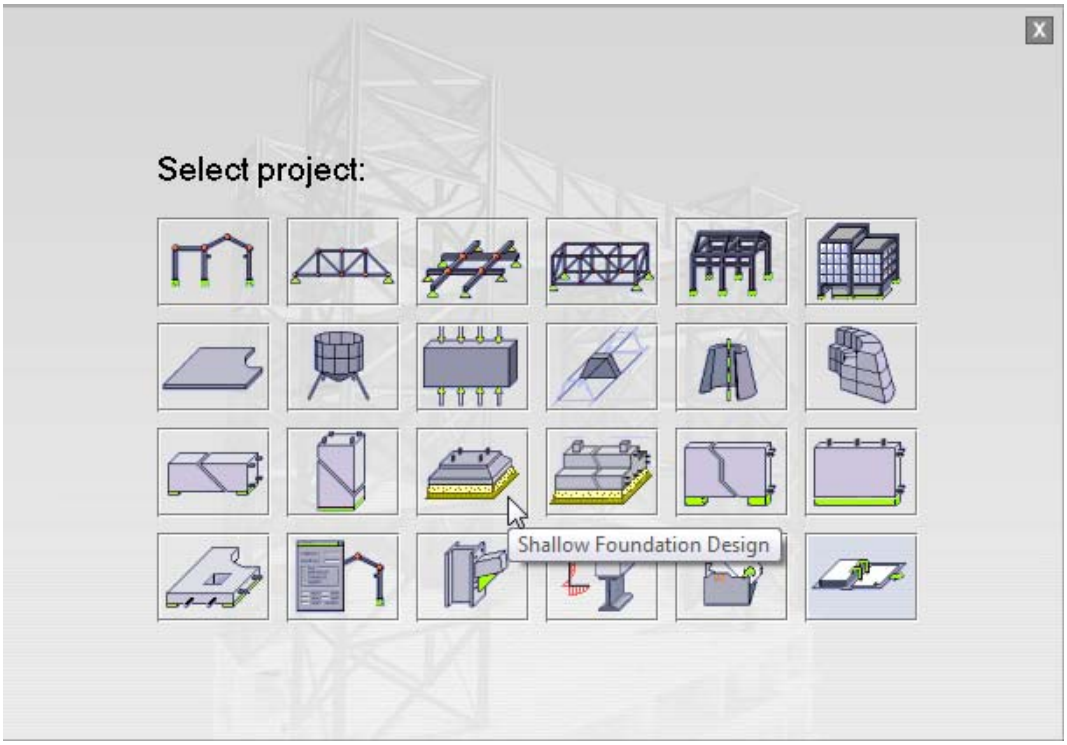
~ Dead load អោយតំលៃ : N = 350 kN , Fx = 50 kN , My = 70 kN m

~ Live load អោយតំលៃ : N = 150 kN , Fx = 30 kN , My = 50 kN m

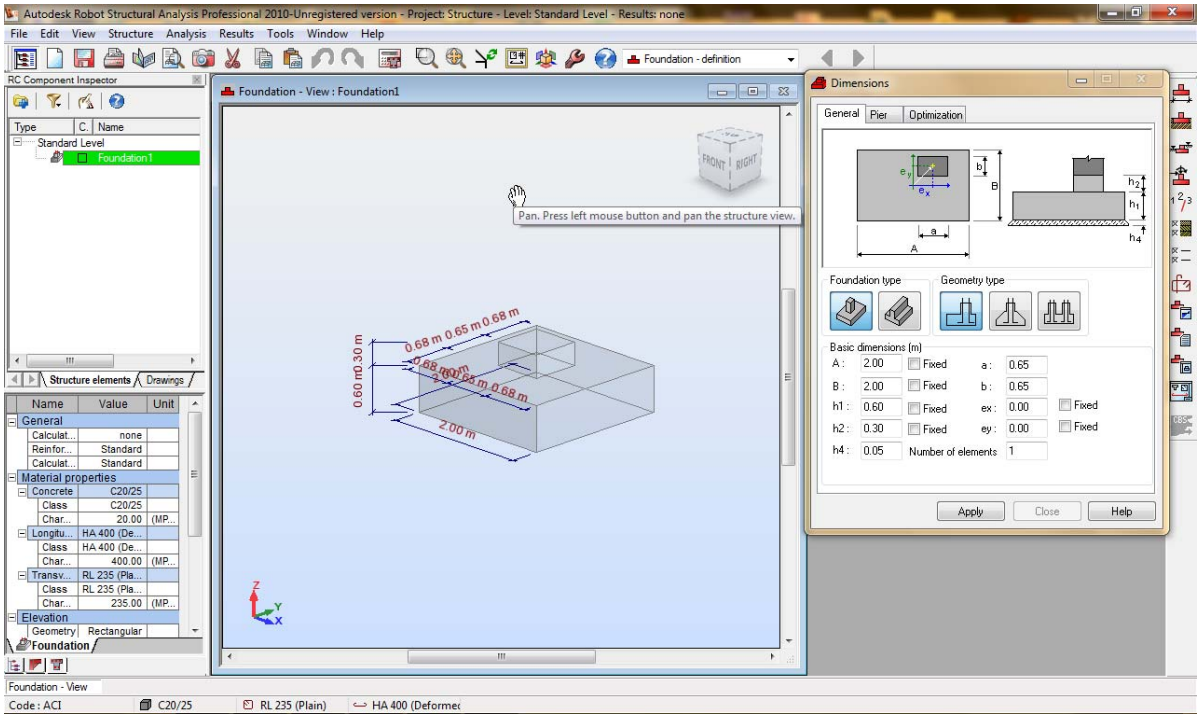
- <u>Analysis and Design ដោយ :</u> ACI code
- <u>Material Properties for Analysis and Design :</u>
  - Yield Strength of Longitudinal Reinforcement : 390 MPa
  - Yield Strength of Transversal Reinforcement : 235 MPa
  - Compressive Strength of Concrete (Cylinder) : 25 MPa

Step by Step Solution

នៅពេលបើក Program Autodesk Robot Structural Analysis Professional 2010 ដំបូងយើងនិងឃើញដូចរូបខាងក្រោមបន្ទាប់មកយើងជ្រើសរើសយក Icon  ដូចដែលបានបង្ហាញនៅលើរូបខាងក្រោម :



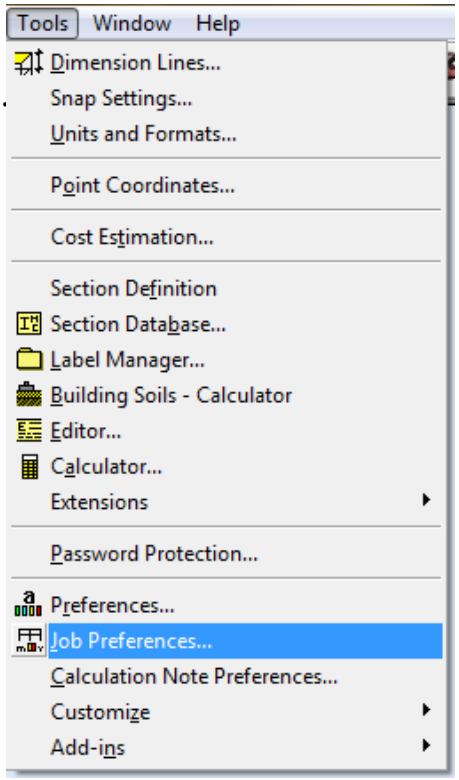
បន្ទាប់មកទៀតអេក្រង់នឹងបង្ហាញដូចរូបខាងក្រោមនេះ



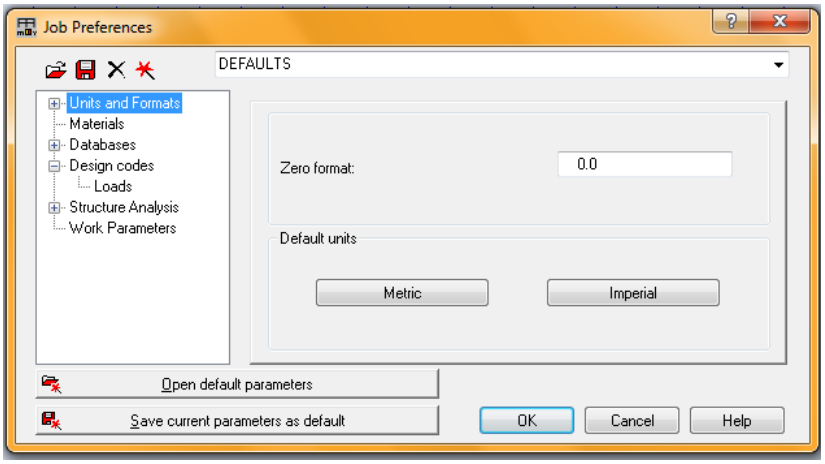
បន្ទាប់មកចូរកំណត់ Units and Format, Materials and Codes មុនពេលធ្វើការ

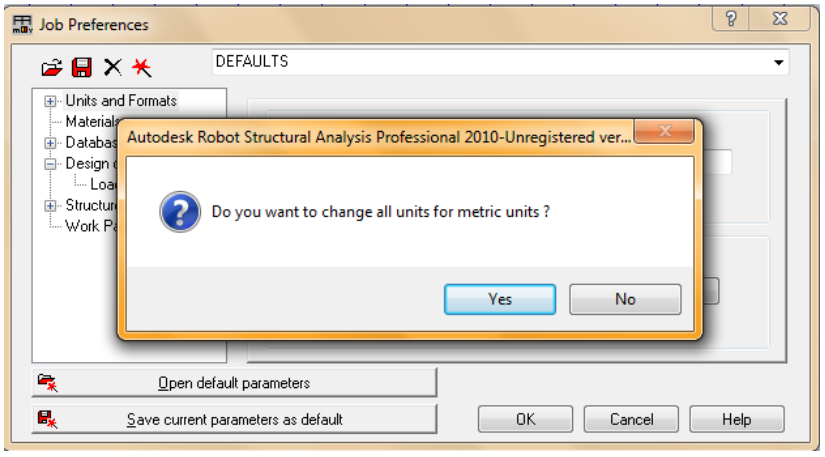
Analysis and Design :

I. ចុចលើ Tool Menu > Job Preference……

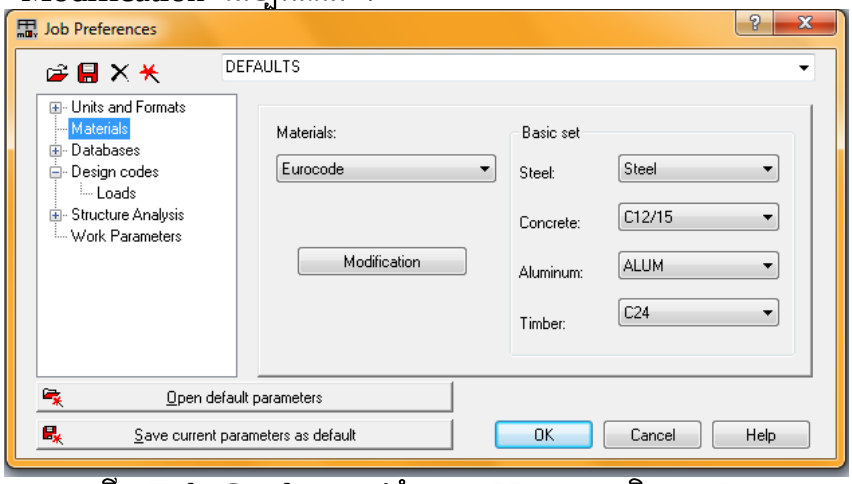


1. អេក្រង់នឹងបង្ហាញផ្ទាំងដូចរូប នៅត្រង់ Units and Formats ត្រូវកំណត់វាជា Metric រួច Yes

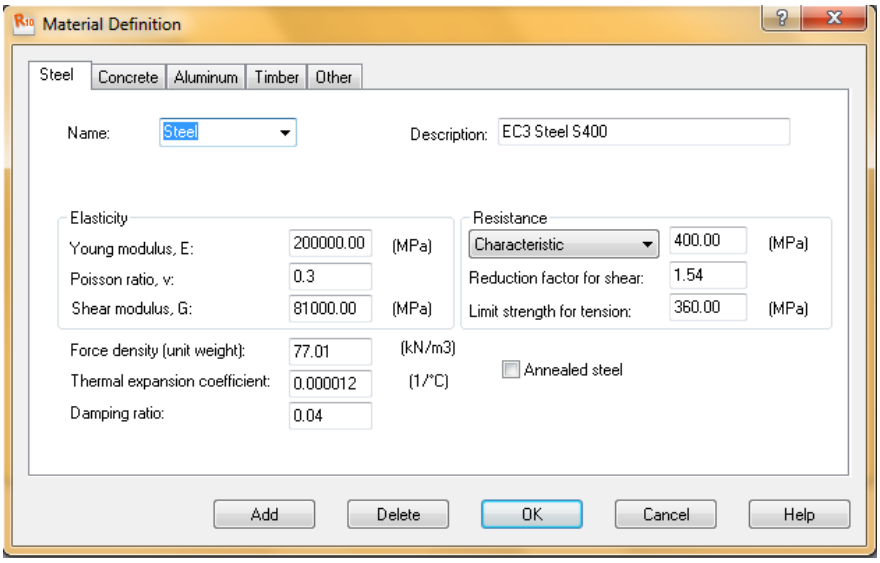




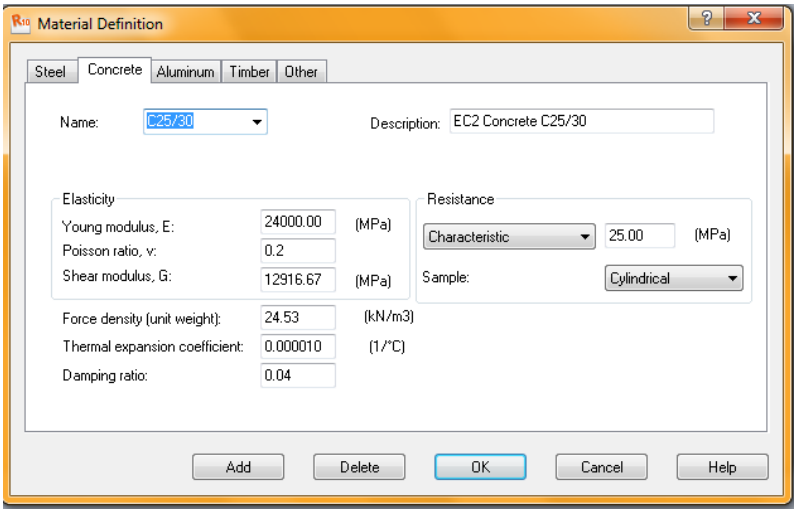
2. នៅត្រង់ Materials ត្រូវកំណត់កូដអោយ Materials ត្រូវយកកូដ Euro code រួចចុចទៅលើ Modification ដើម្បីកំណត់ ។



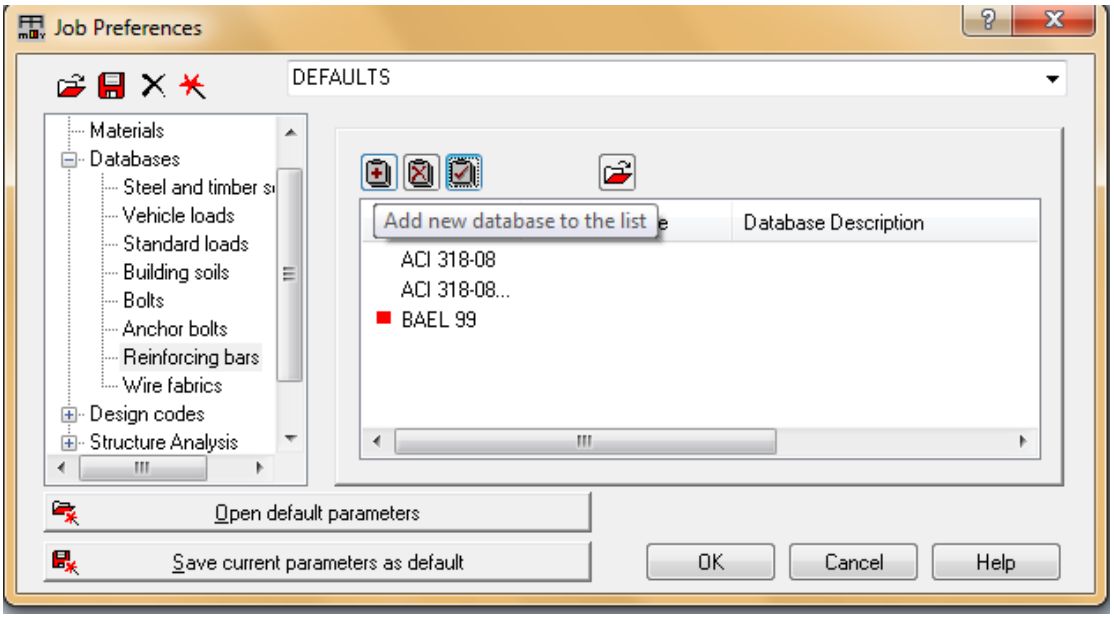
3. រួចចុចលើ Tab Steel រួចត្រូវបំពេញ Name និង Description ដូចខាងក្រោម

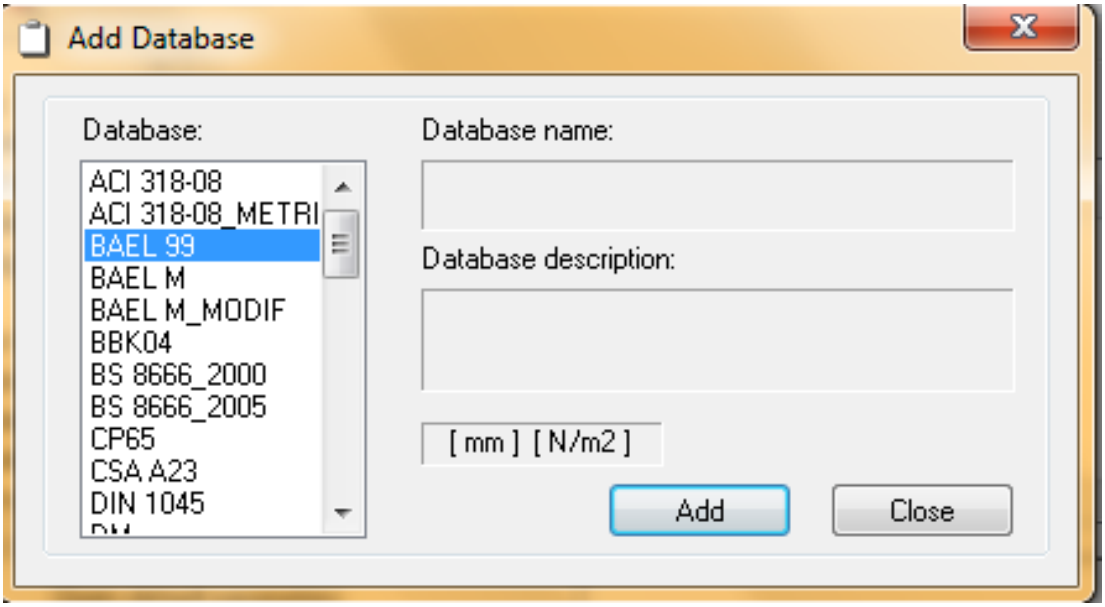


4. រួចចុចលើ Tab Concrete រួចត្រូវបំពេញ Name និង Description និង Young modulus ដូចខាងក្រោម រួចចុច OK ។

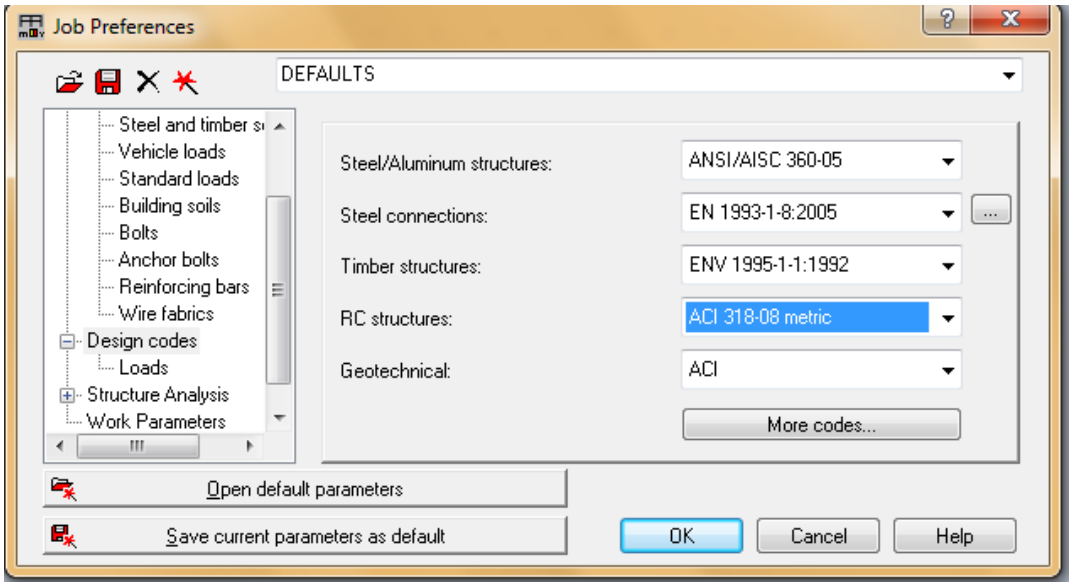


5. រួចចុចលើ Databases រួចរើសយក Reinforcing bars ហើយនៅប្រអប់ខាងស្តាំ ត្រូវបង្កើតកូដ BAEL99 ដោយចុចទៅលើ Icon ដែលមានឈ្មោះថា Add new databases to the list នោះវានឹងចេញតារាង Add Databases រួចរើសយក BAEL99 រួចចុច add ។ បន្ទាប់មករើសកូដដោយយក Mouse select on code រួចចុច Icon ដែលមានឈ្មោះថា Set as the current database ។

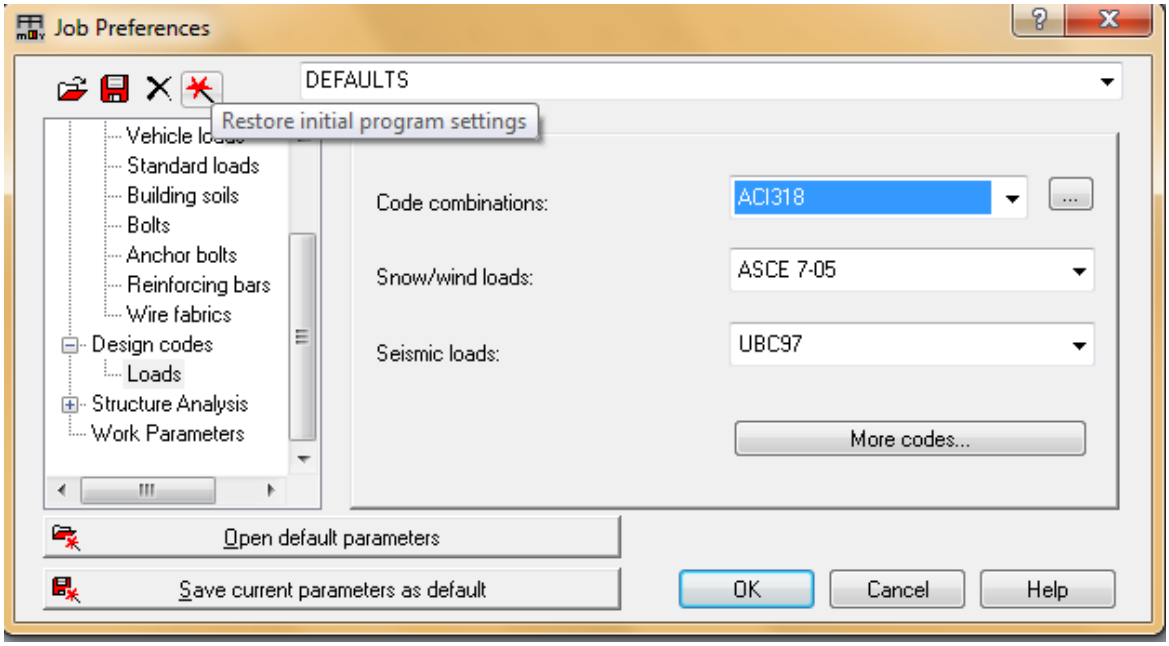




6. រួចចុចលើ Design codes ក្នុងប្រអប់ RC Structures ត្រូវកំណត់ជា ACI 318-08 metric

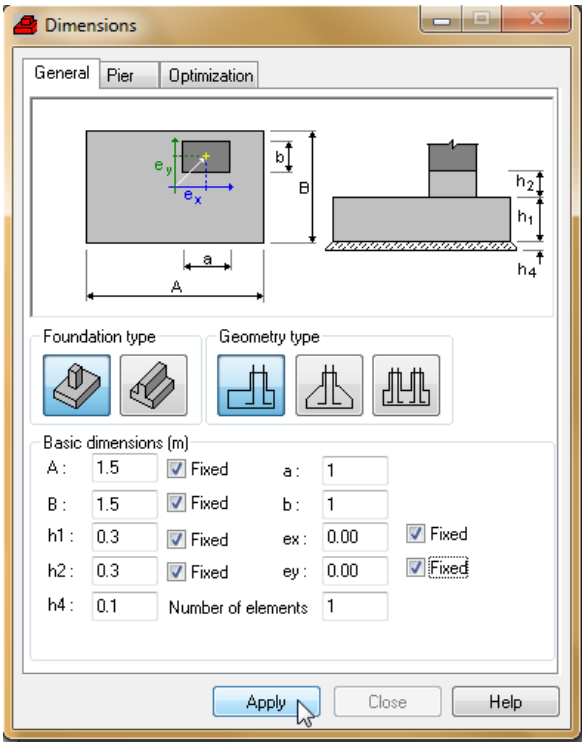


7. នៅត្រង់ Loads ក្នុងប្រអប់ Code combinations ត្រូវកំណត់ជា ACI318 រួចចុច OK ។



ចំនាំ: បើសិនជាយើងចង់ប្រើខ្នាត និង កូដដែលយើងកែប្រែខាងលើជារឿងរហូត ដោយយើងមិនចាំបាច់កែប្រែវានៅរឿងរាល់បើកកម្មវិធីម្តងៗនោះ បន្ទាប់ពីយើងចុច ok ហើយនោះយើងត្រូវចុច Save job Preferences បន្ទាប់មកយើងដាក់ឈ្មោះថា ACI318-08Metric រួចចុច Save រួចហើយចុច Save current parameters as default នោះឈ្មោះ ACI318-08Metric នឹងលោតចូលទៅបន្ថែមនៅក្នុង DEFAULTS នោះយើងនឹងបានខ្នាត និង កូដ ACI318-08Metric នៅរឿងរាល់ពេលបើកកម្មវិធីម្តងៗ ។

II. ការកំណត់ទំហំគ្រឹះ សសរ និង ប្រភេទគ្រឹះ (Foundation Definition)



១. ការកំណត់ទំហំជើងតាង និងសសរគឿ នៅក្នុង Tab General -វាយកំណត់តំលៃដូចរូបខាងក្រោម រួចហើយចុច Apply ។

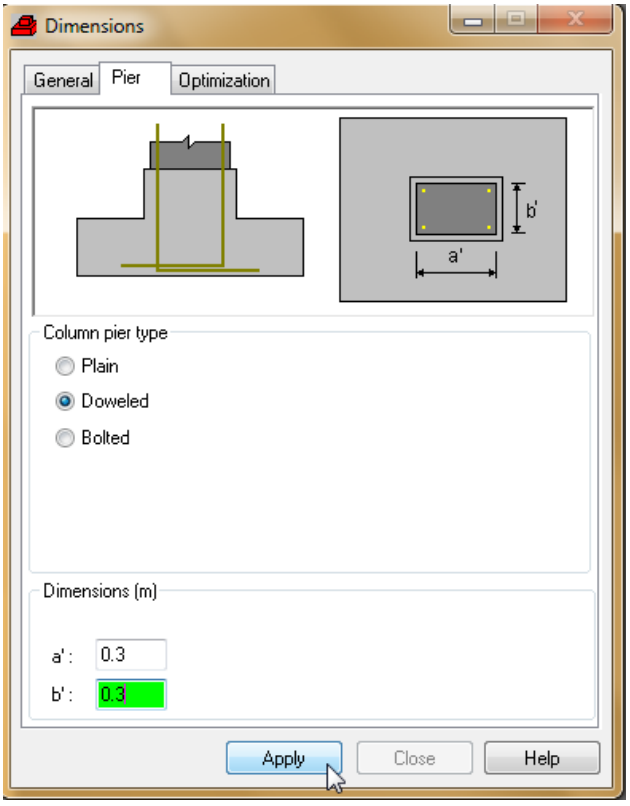
Foundation Type : ជាប្រភេទគ្រឹះជើងតាងសំរាប់កំណត់រូបរាង រឺលក្ខណៈជើងតាង ។

Basic dimensions : សំរាប់កំណត់ទំហំ និងកំរាស់ជើងតាងនិង សសរគឿ ។

-បើ Tick យក Fixed មានន័យថាឱ្យ Program យកតំលៃដែលកំណត់ទៅធ្វើការគណនា ។ បើសិនមិន Tick យក Fixed មានន័យថាឱ្យ Program ជ្រើសរើសទំហំជើងតាងឱ្យដោយគោរពលក្ខណៈនៅក្នុង Tab Optimization ។

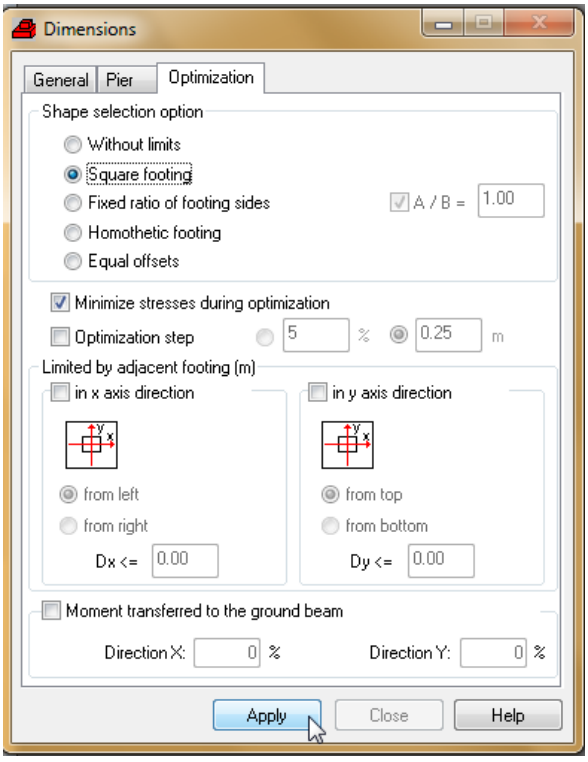
២. <u>ការកំណត់ទំហំសសរ Tab Pier</u>

a' = 0.3m , b'= 0.3m ជាមុខកាត់សសរតពីសសរគឿ ជើងតាងឡើង។ រើសយក Doweled រួចចុច Apply ។



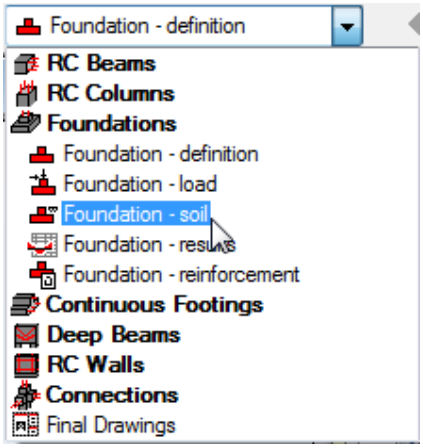
៣. <u>ការកំណត់លក្ខណៈដើម្បីឱ្យ Program ជ្រើសរើសទំហំជើងតាងឱ្យ Tab Optimization</u>

ដោយជើងតាងជាការត្រូវយក Square footing រួចចុច Apply ។



III. ការកំណត់ស្រទាប់ដីនៅពីក្រោមគ្រឹះ (Foundation Soil)

១. ដំបូងចុចលើ Icon  នៅខាងស្តាំអេក្រង់រីចុចលើ Layout ហើយរើសយក Foundation soil កន្លែងដែលត្រូវបំពេញ :

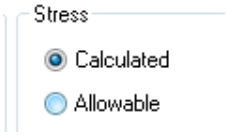


- Backfill heights N_1 = 0.00

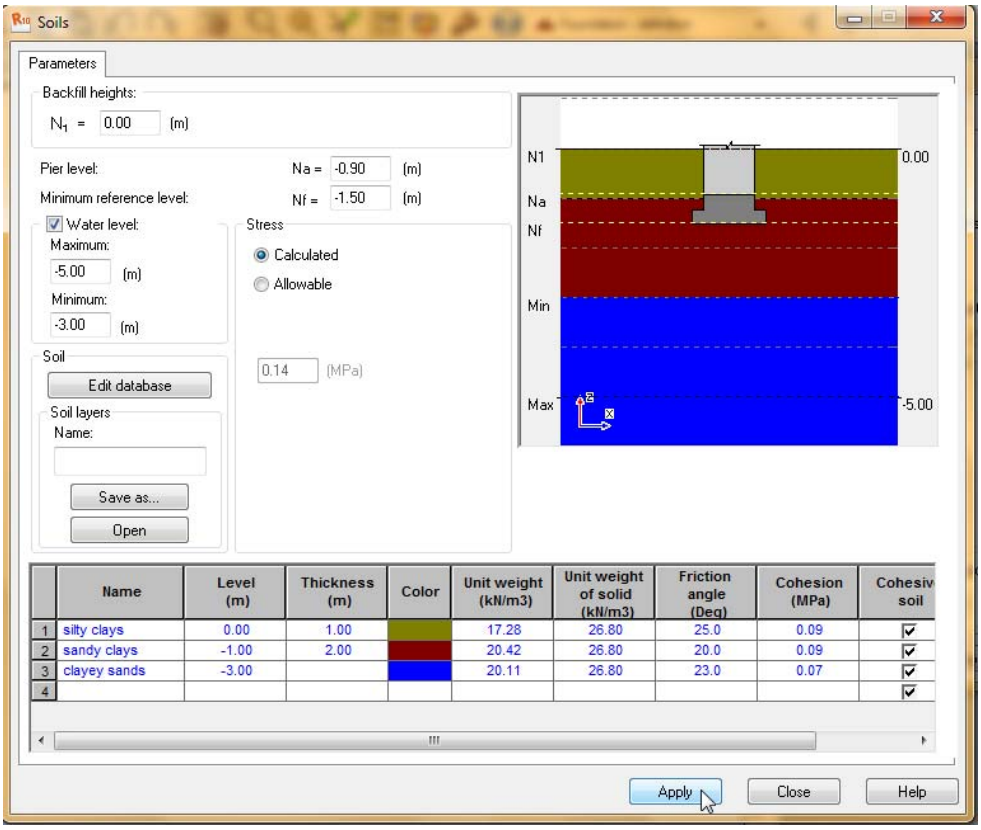
-Pier Level : N_a =-0.90 , N_f = -1.50

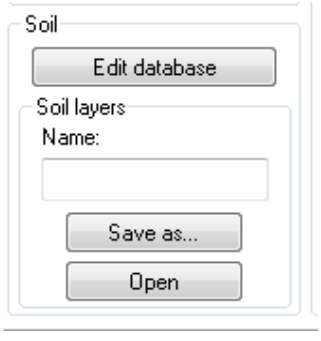
-Ground water level : Maximum = -5.00m , Minimum = -3.00m

-Click កន្លែង Stress យក Calculated



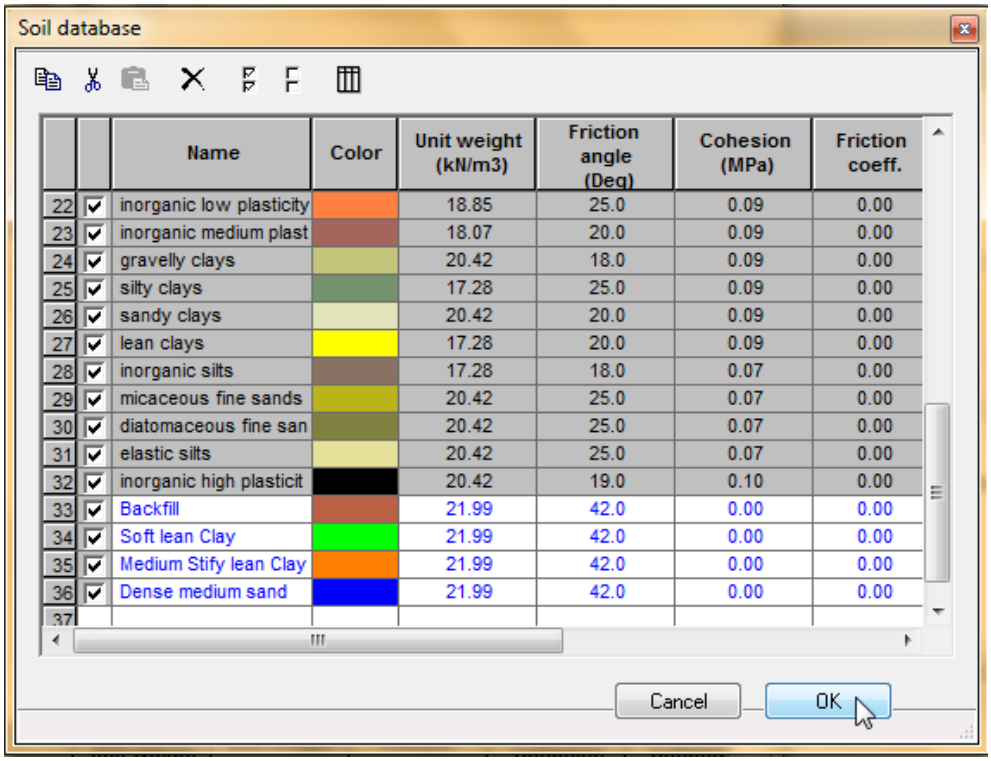
-ជ្រើសរើសប្រភេទដី បង្ហាញដូចរូបភាពខាងក្រោម :





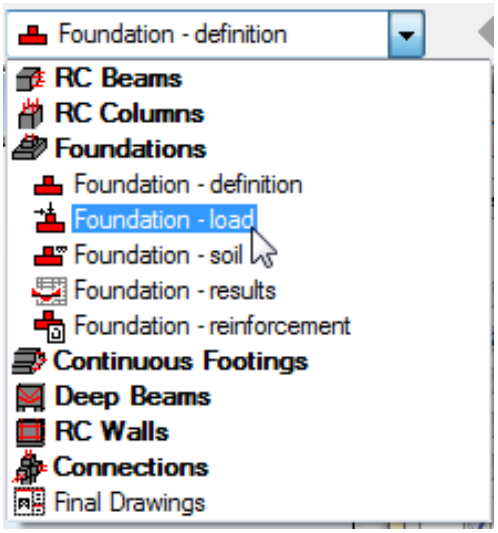
២. ដើម្បីបង្កើតឈ្មោះដីដែលយើងចង់បានបន្ថែម :

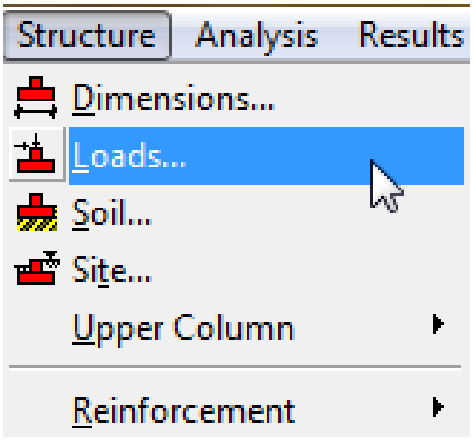
ដើម្បីកំណត់ស្រទាប់ដី ត្រូវចុចលើ Edit database បន្ទាប់មកចុចលើ ចុចលើ Cell ចុងក្រោយគេដែលស្ថិតនៅក្នុង Column “ Name ” រួចវាយឈ្មោះដី និង តំលៃលទ្ធផលដីដូចតារាងក្នុងរូបខាងក្រោម រួចចុចលើពាក្យ ok ។

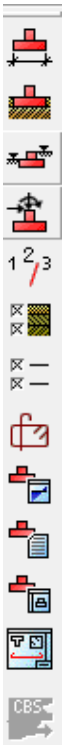


IV. ការកំណត់បន្ទុកដែលមានអំពើមកលើគ្រឹះ (Foundation Load)

១. ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំអេក្រង់រឺចុចលើ Structure > Loads…. ម្យ៉ាងទៀតចុចលើ Layout ហើយរើសយក Foundation load .







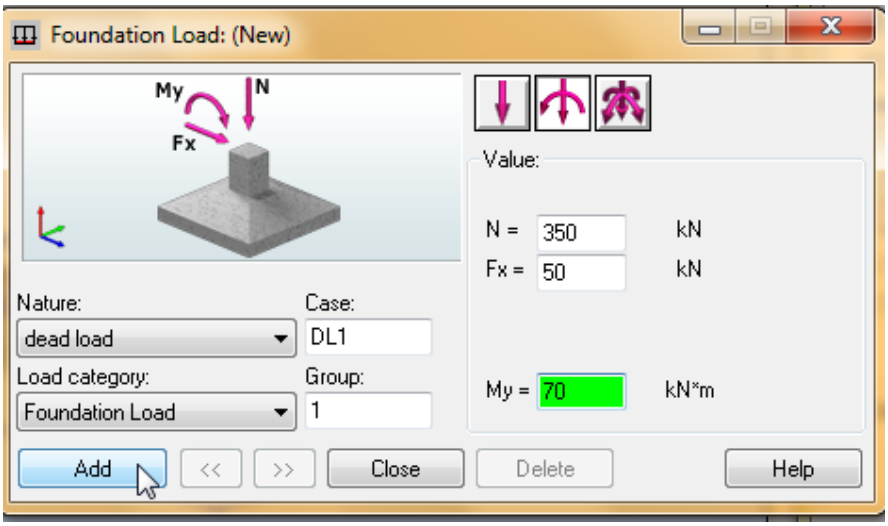
២. ការកំណត់បន្ទុកថេរ (Dead Load)

-ចុចលើ Icon រូបនេះ  ហើយ

-ក្នុងប្រអប់ Nature ចុចរើសយក dead load

-ក្នុងប្រអប់ Load category ចុចរើសយក Foundation load

-វាយតំលៃបន្ទុក Dead load ដែលមាន : កំលាំងចំផ្ចិត N= 350kN កំលាំងបុកតាមទិសដេក Fx = 50kN និង ម៉ូម៉ង់ My=70kNm រួចហើយចុច Add ។



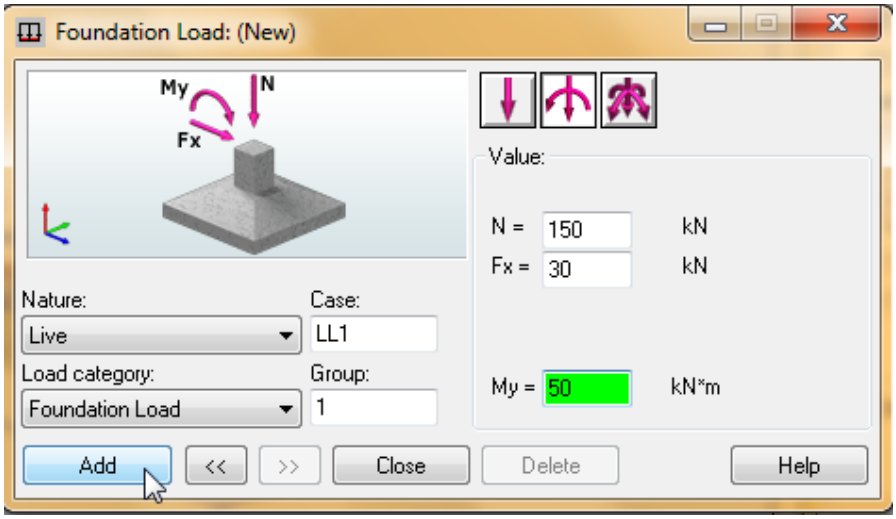
៣. ការកំណត់បន្ទុកអថេរ (Live Load)

-ចុចលើ Icon រូបនេះ  ហើយ

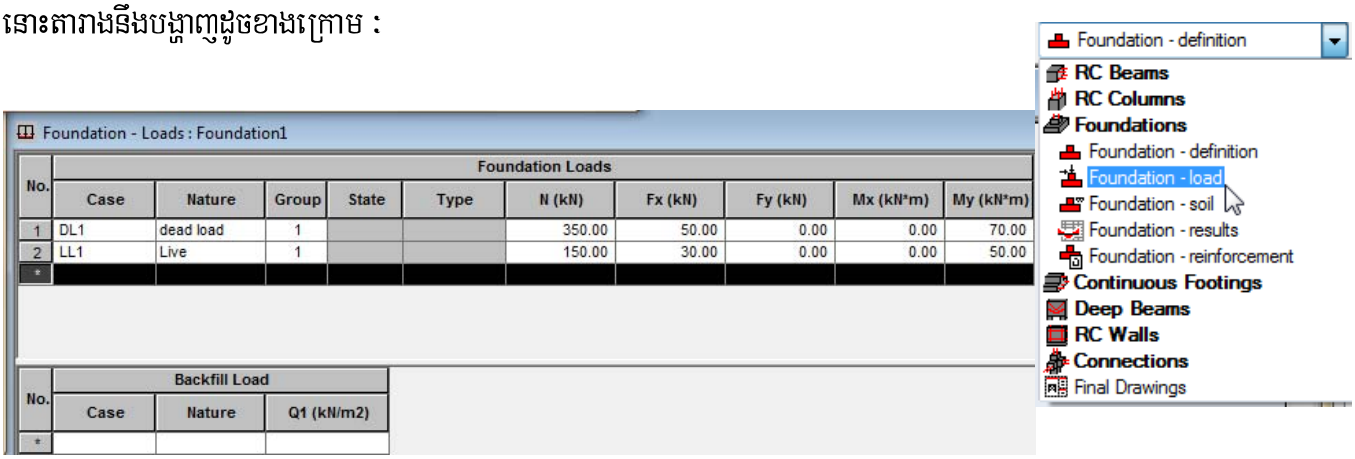
-ក្នុងប្រអប់ Nature ចុចរើសយក live load

-ក្នុងប្រអប់ Load category ចុចរើសយក Foundation load

-វាយតំលៃបន្ទុក Live load ដែលមាន : កំលាំងចំផ្ចិត N= 150kN កំលាំងបុកតាមទិសដេក Fx = 30kN និង ម៉ូម៉ង់ My=50kNm រួចហើយចុច Add និង Close ។



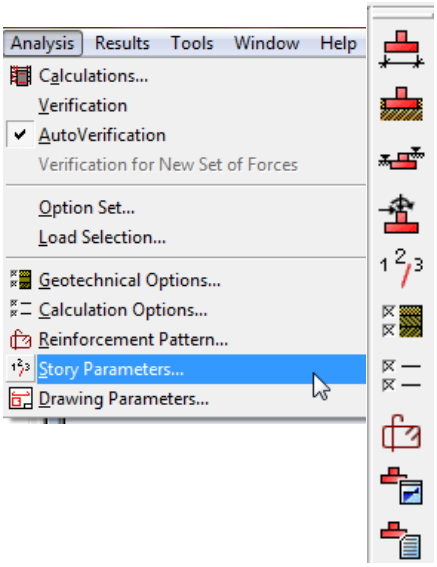
-បន្ទាប់មកយើងអាចមើលតារាងសង្ខេបបន្ទុកដែលមានអំពើមកលើគ្រឹះជើងតាង ដោយចុចលើ Layout រើសយក Foundation load នោះតារាងនឹងបង្ហាញដូចខាងក្រោម :

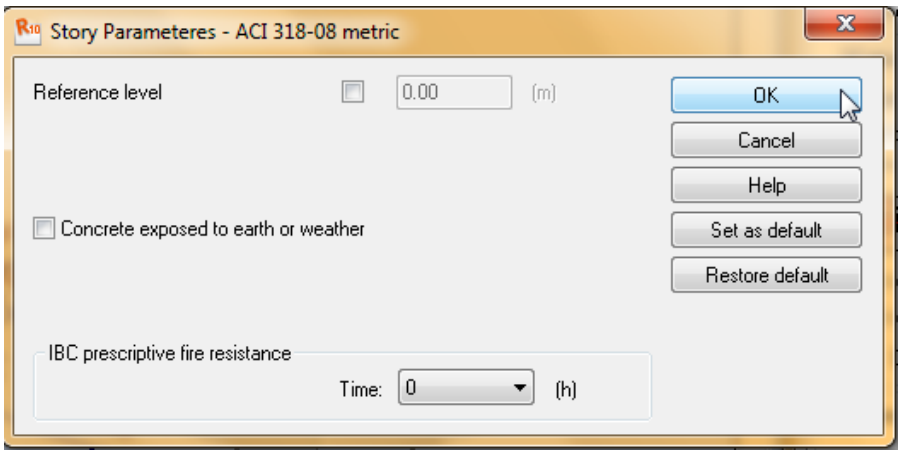


V. **ការកំណត់** (Story Parameters) ដូចរូប

-ដំបូងចុចលើ Icon រូបនេះ  នៅខាងស្តាំអេក្រង់រឺចុចលើ Analysis Menu > Story Parameters ....

នោះនឹងចេញផ្ទាំងដូចខាងក្រោម :



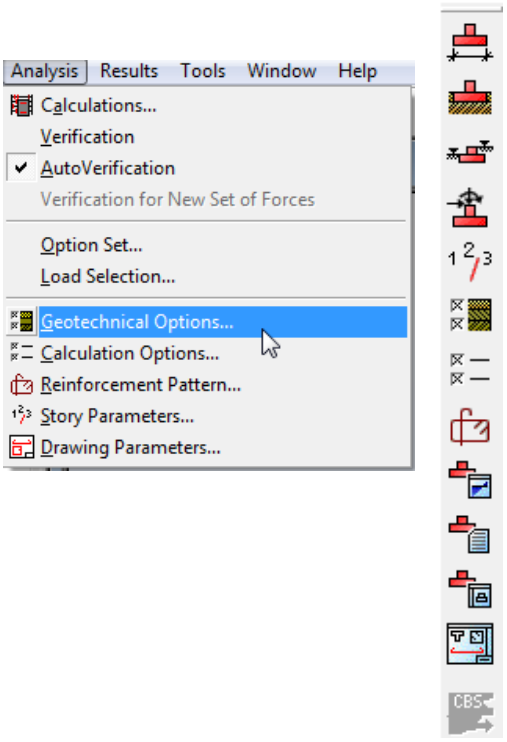


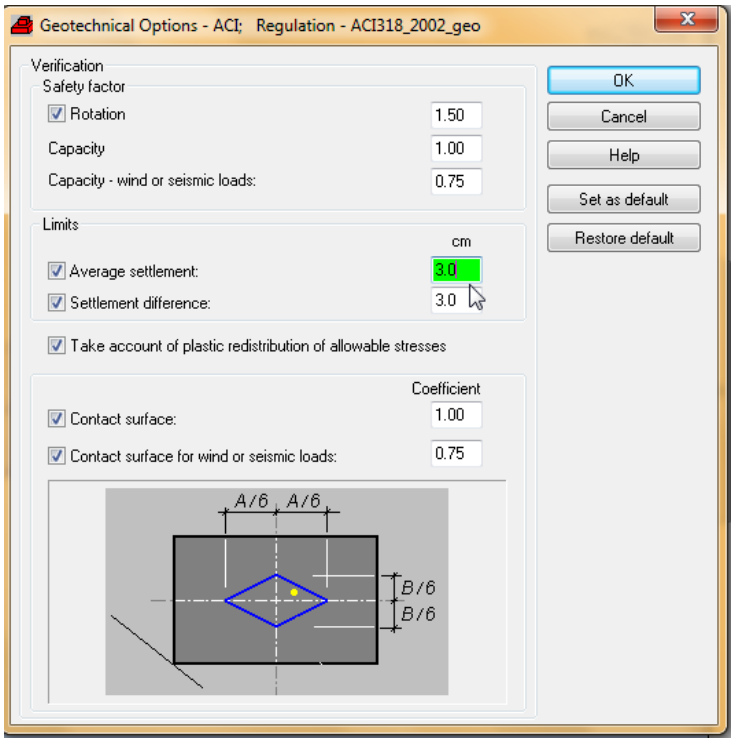
VI. **ការកំណត់** (Geotechnical Option) ដូចរូប

-ដំបូងចុចលើ Icon រូបនេះ  នៅស្តាំអេក្រង់រឺចុចលើ Analysis Menu > Geotechnical Option ...

នៅត្រង់កន្លែង Limits សំរាប់កំណត់សំរុត :

Average settlement : សំរុតមធ្យម

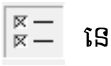


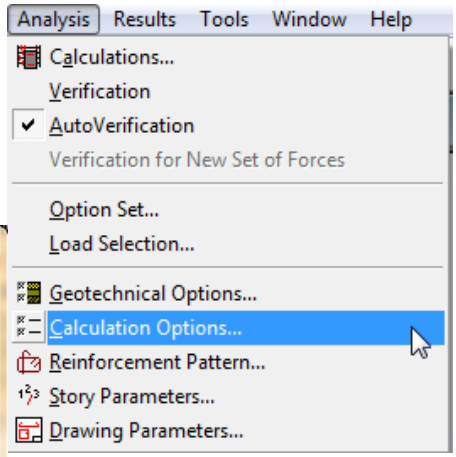


VII. **<u>ការកំណត់</u>** (Calculation Options) ដូចរូប

នៅក្នុង Calculation Option មាន ៤ Tab ដែលត្រូវកំណត់

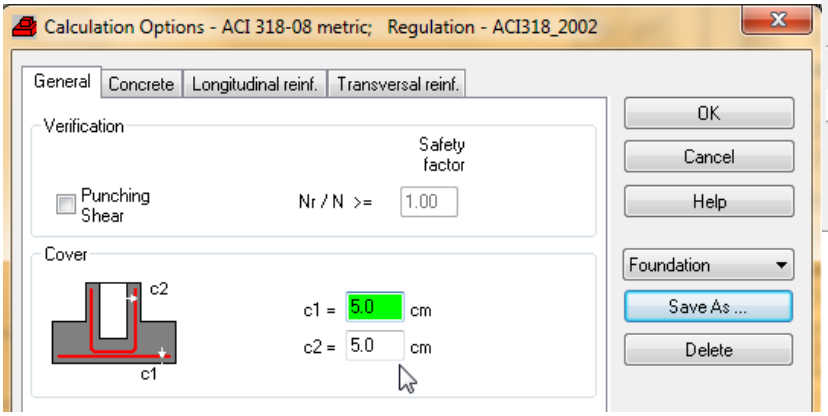
- Tab General : កំណត់ស្រទាប់ការពារ គិតពីប្រវែងសសរចូលគណនា និងការកំណត់ផ្សេងទៀត.....
- Tab Concrete : កំណត់រ៉ែស៊ីស្តង់ សង្កត់របស់បេតុងនៅអាយុ២៨ថ្ងៃ
- Tab Longitudinal reinf : កំណត់រ៉ែស៊ីស្តង់ដែកបណ្តោយ និងអង្កត់ផ្ចិត
- Tab Transversal reinf : កំណត់រ៉ែស៊ីស្តង់ដែកកង និងអង្កត់ផ្ចិត

ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំងអេក្រង់ រឺចុចលើ Analysis Menu > Calculation Option ……

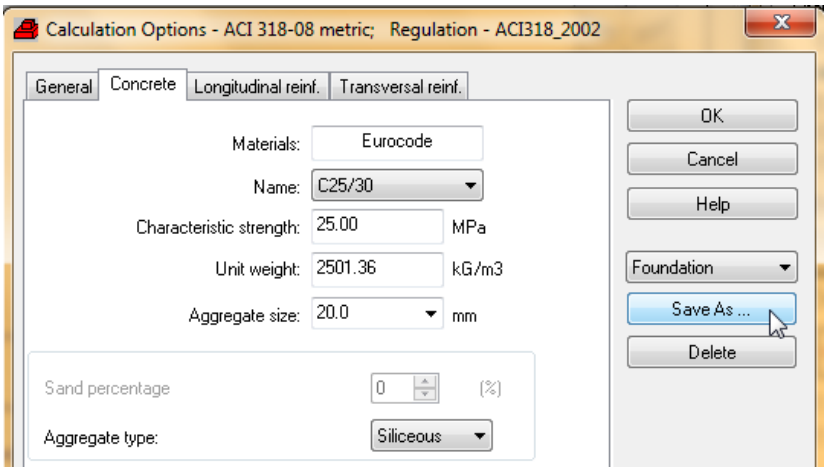


នោះនឹងចេញផ្ទាំងដូចខាងក្រោម :

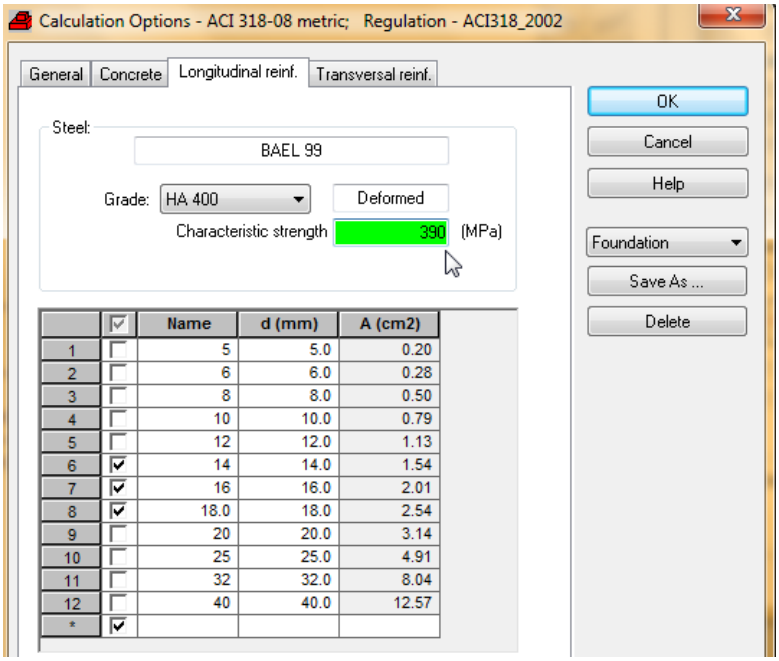
<u>Tab General:</u>



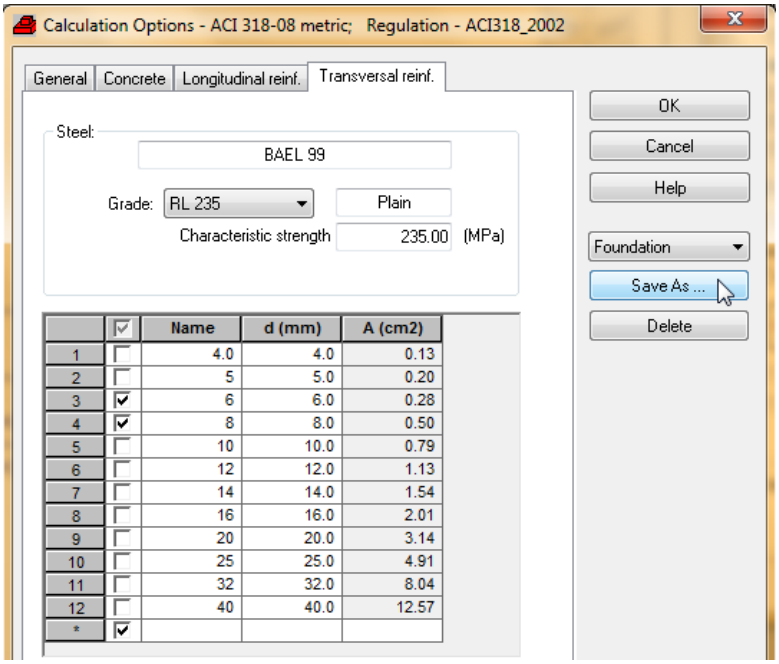
<u>Tab Concrete :</u>



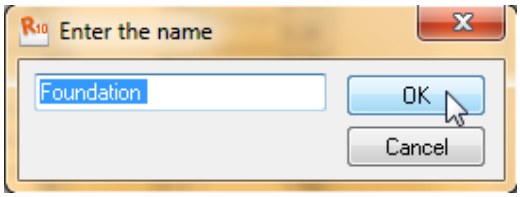
Tab Longitudinal reinf ..



Tab Transversal reinf ….



-ពេលកំណត់រួចរាល់ហើយ ចុចលើ Save as ….. រាយឈ្មោះ Foundation រួចចុច ok ដើម្បីរក្សាទុក ។

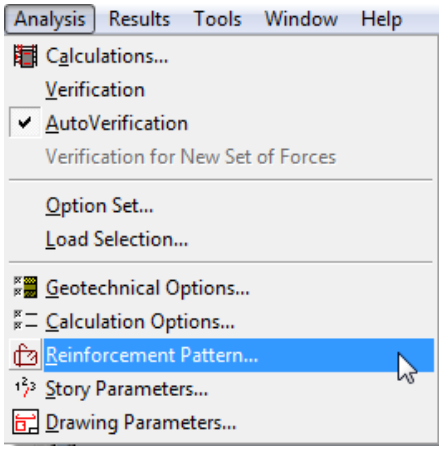


VIII. **<u>ការកំណត់</u>** (Reinforcement Pattern) ដូចរូប

នៅក្នុង <u>Reinforcement Pattern</u> មាន ៤ Tab ដែលត្រូវកំណត់

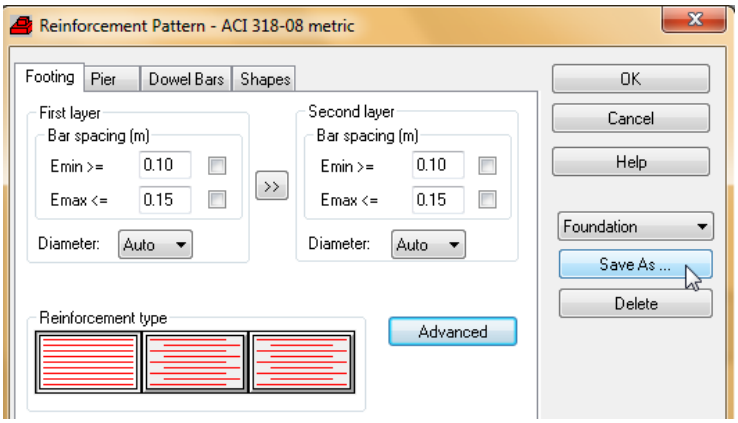
- Tab Footing bars : កំណត់គំលាតពីសសៃដែក១ទៅសសៃដែក១ ការចងដែក.....
- Tab Pier bars : កំណត់គំលាតចងដែកកង និងការពត់ដែកកង .....
- Tab Dowel bars : កំណត់ប្រវែងដែកសំរាប់ទុកអោយតភ្ជាប់ទៅសសរខាងលើ.....
- Tab Shapes : កំណត់ទ្រង់ទ្រាយ លក្ខណៈនៃការចងដែក.....

ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំអេក្រង់ រីចុចលើ Analysis Menu > Reinforcement Pattern .....

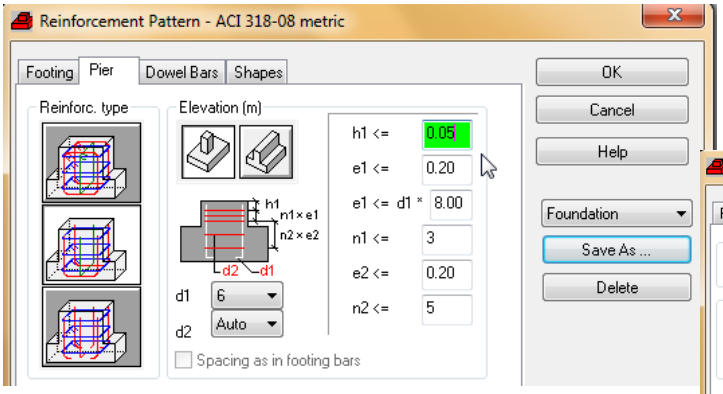


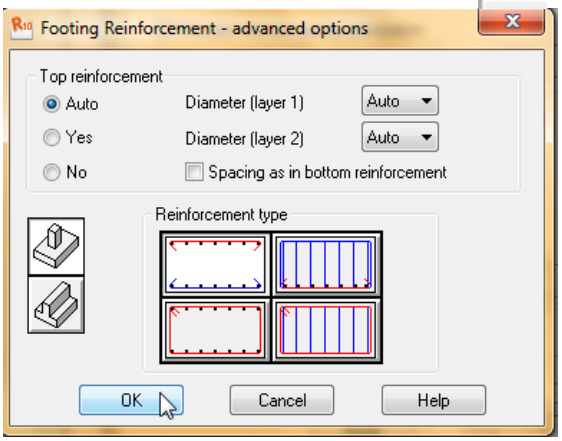
នោះនឹងចេញផ្ទាំងដូចខាងក្រោម :

Tab Footing bars:

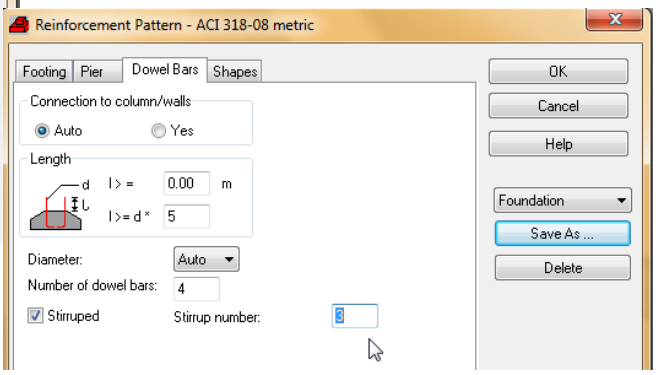


Tab Pier bars :



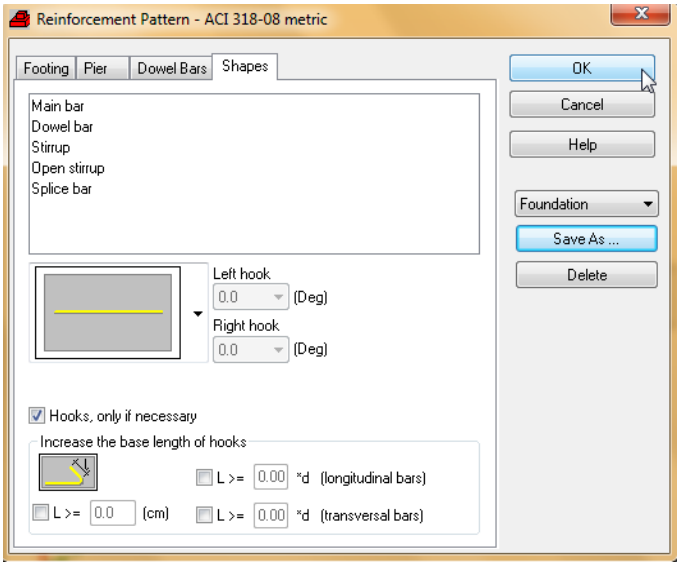


Tab Dowel bars :

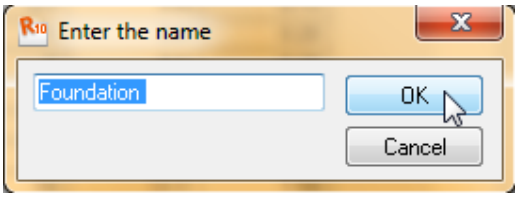


Stirrup number : ចំនួនដែកកងរងចាំ

Tab Shapes :



ពេលកំណត់រួចរាល់ហើយ ចុចលើ Save as ….. រាយឈ្មោះ Foundation រួចចុច ok ដើម្បីរក្សាទុក ។



IX. ការចាប់ផ្តើមគណនារកសសៃវែក (Start Calculation) ដូចរូប

ដំបូងចុចលើ Icon  នេះនៅខាងលើអេក្រង់ រឺចុចលើ Analysis Menu > Calculation …….

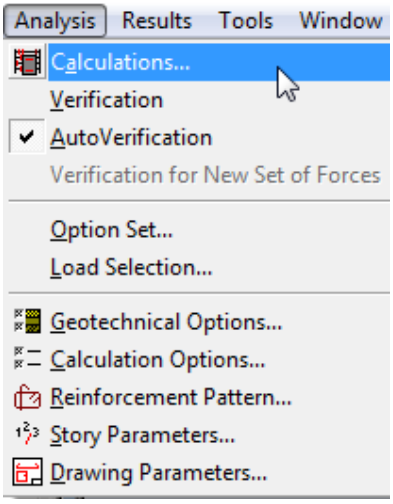


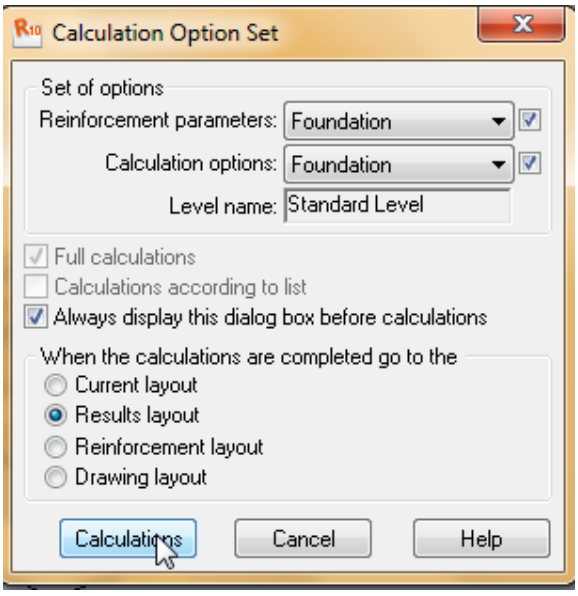
នោះនឹងចេញផ្ទាំងដូចខាងក្រោម : បន្ទាប់មកនៅត្រង់កន្លែង

Reinforcement parameters : ជ្រើសរើសយកឈ្មោះដែលយើងបាន Save ទុក (Foundation)

Calculation options : ជ្រើសរើសយកឈ្មោះដែលយើងបាន Save ទុក (Foundation) រួចចុច

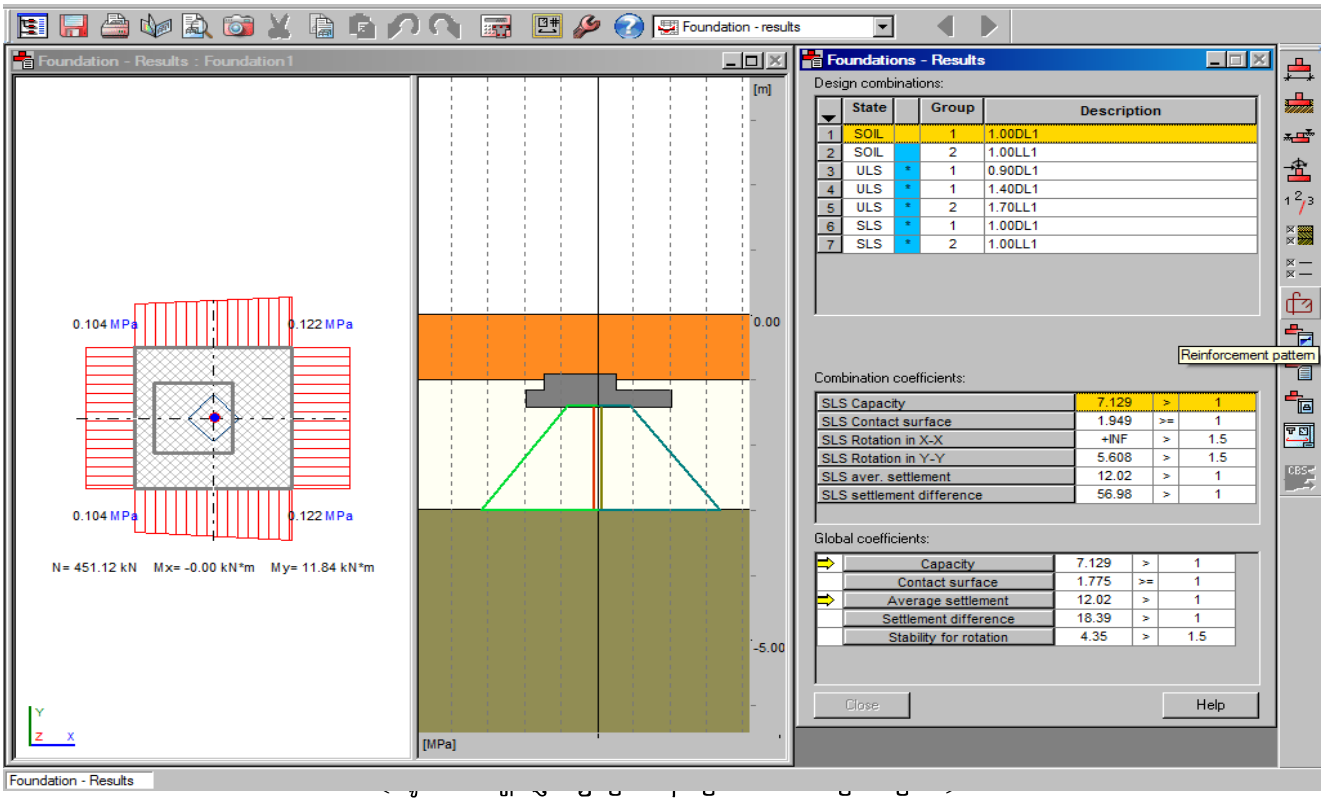
Results Layout ដើម្បីបង្ហាញលទ្ធផល រួចចុច Calculation ។





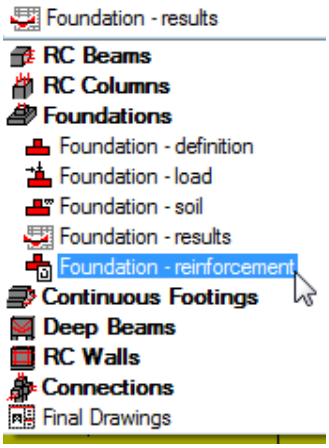
ជ្រើសរើសឈ្មោះត្រង់ Reinforcement Parameters និង Calculation Options ដែលយើងបានកំណត់រួចមក ហើយនៅចំនុចទី ៧ និងទី ៨ ខាងលើគឺឈ្មោះថា Foundation ។

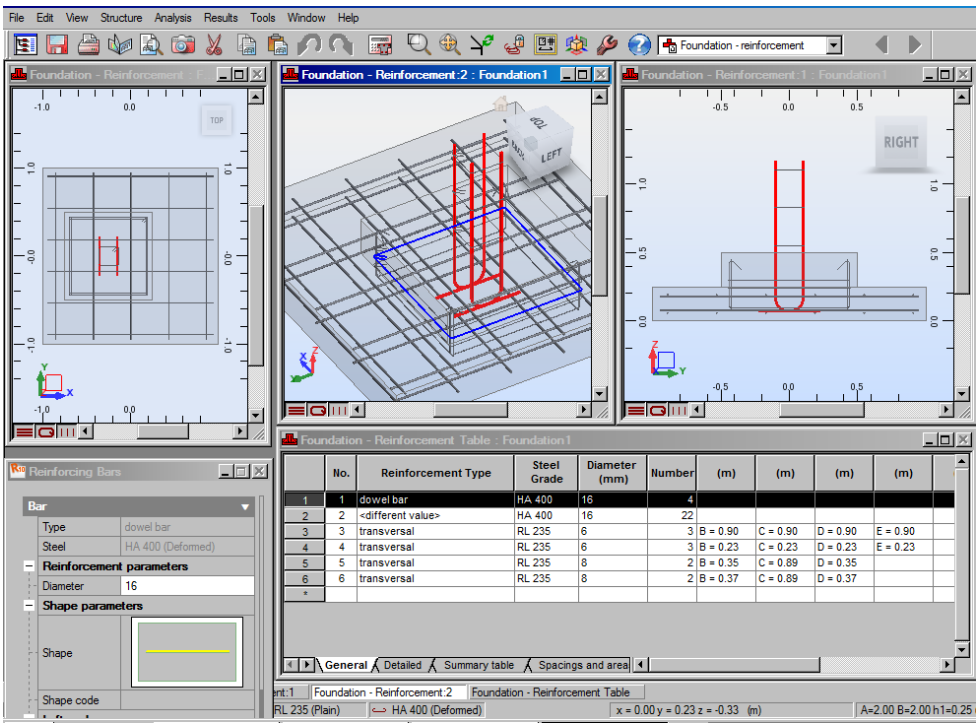
ករណីបើមាន Error & Warning វានឹងបង្ហាញផ្ទាំង Reinforced Concrete បន្ទាប់ពីចុច Calculations រួច ។
បន្ទាប់ពីចុច Calculatons រួចនោះវានឹងចេញផ្ទាំងលទ្ធផលដូចខាងក្រោម : តារាងនេះគ្មានការ Error ទេ ។



X. ការគ្រួតពិនិត្យមើលការដាក់រៀបចំសែកែក (Foundation Reinforcement)

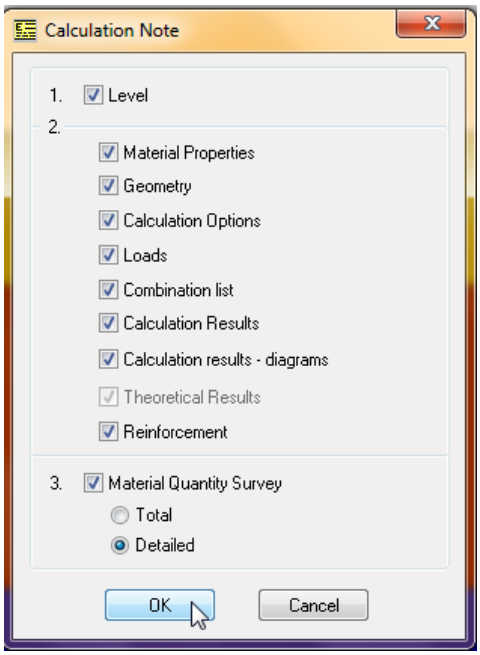
-ចុចលើ Layout ហើយរើសយក Foundation - reinforcement ដូចរូប

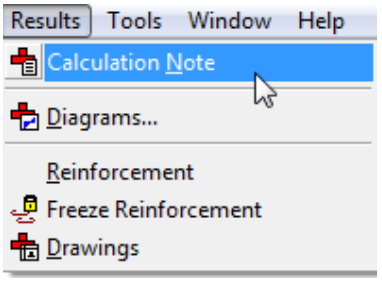


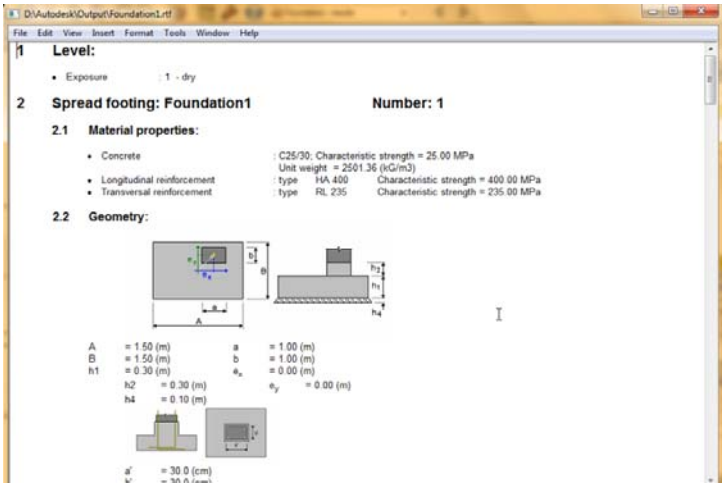


XI. (Calculations Note)

-ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំងអេក្រង់រីចុចលើ Results Menu > Calculation Note បន្ទាប់ពីចុច ok នៅលើផ្ទាំង Calculation Note រួចនឹងចេញផ្ទាំងតំលៃដូចនៅក្នុងរូប ។



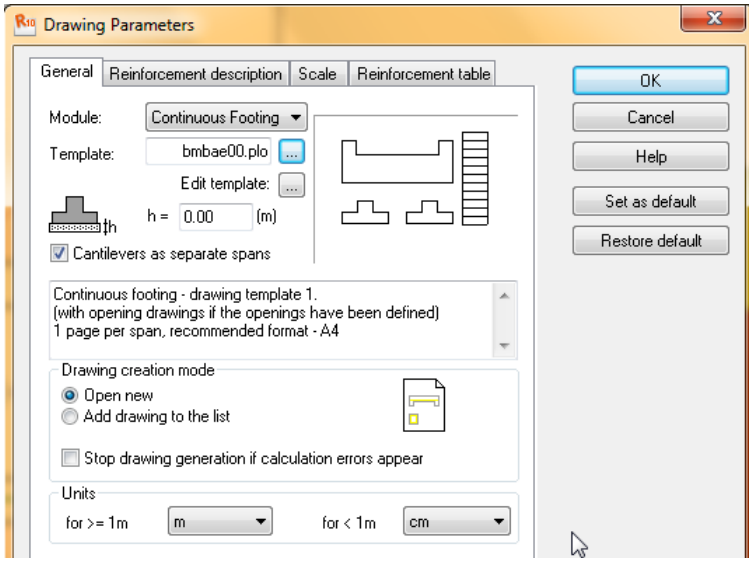


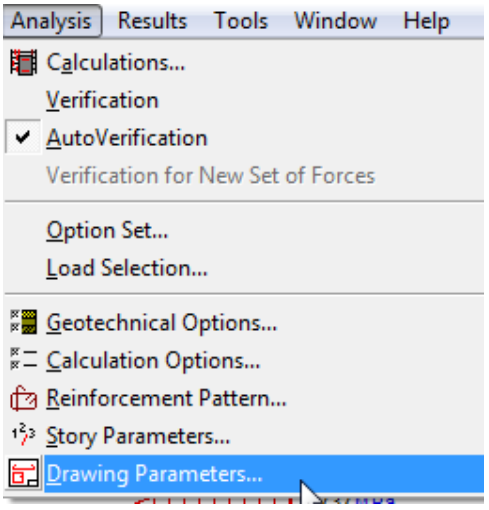


XII. (Drawing Parameters)

-ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំងអេក្រង់រីចុចលើ Analysis > Drawing Parameter ..

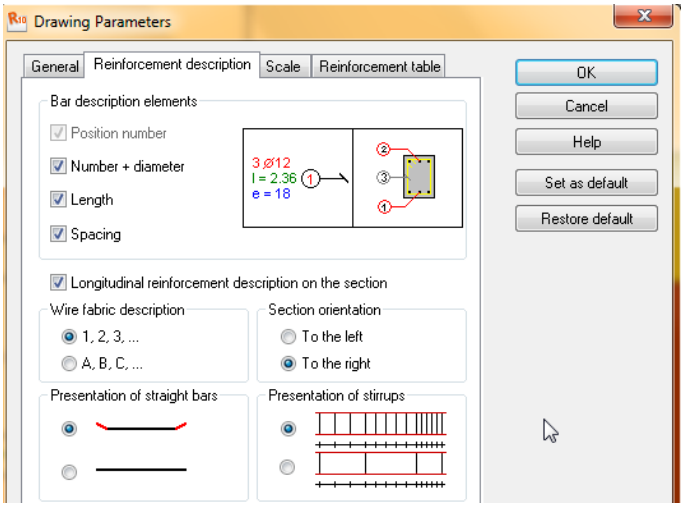
Tab General :

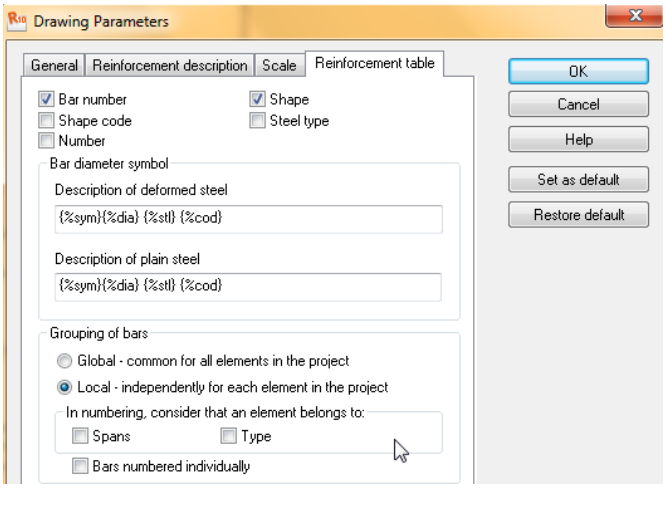






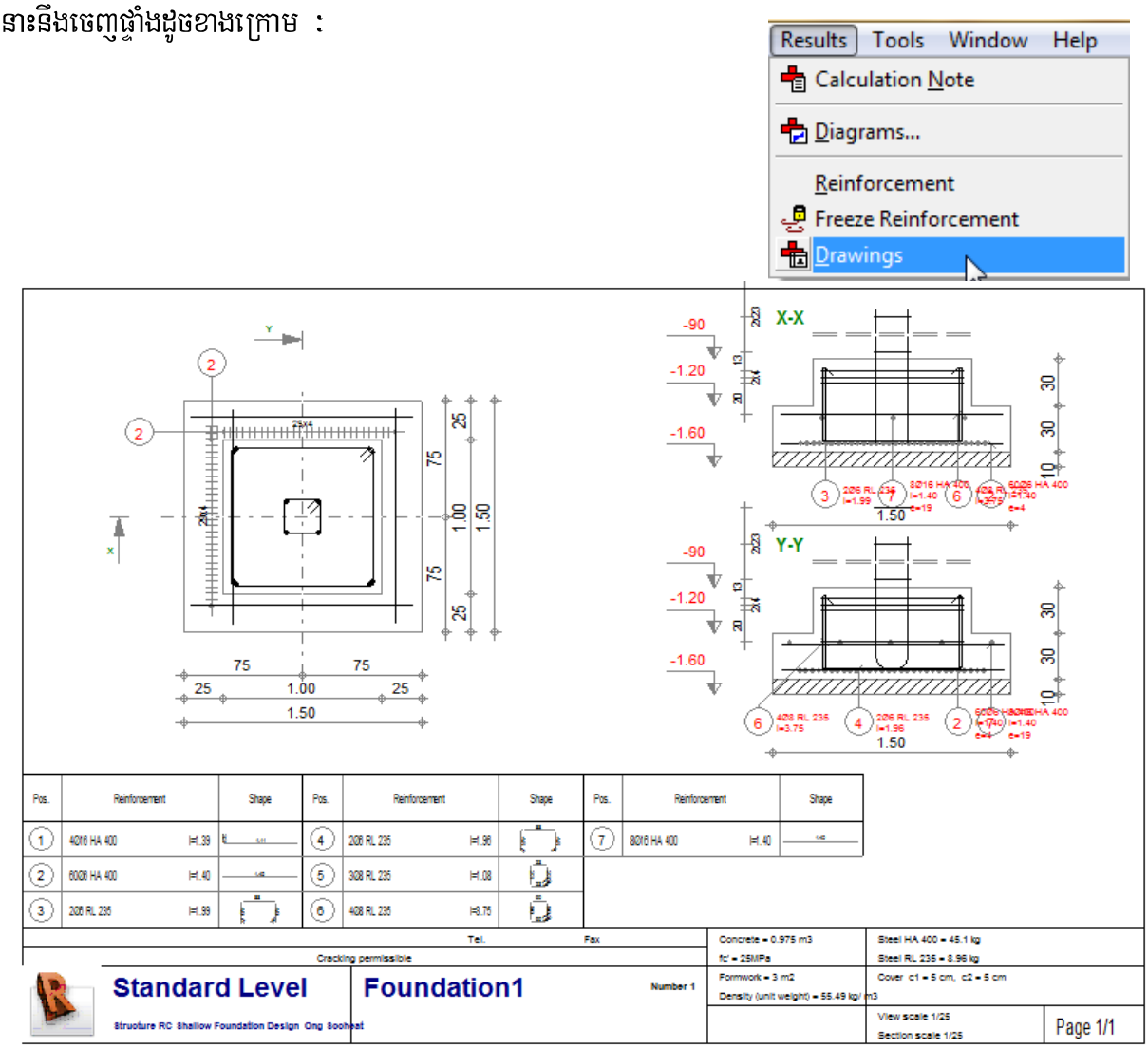
Tab Reinforcement description Tab Scale :





XIII. (មើលបង្ហាញ Drawing)

-ដំបូងចុចលើ Icon  នេះនៅខាងស្តាំងអេក្រង់រីចុចលើ Result Menu > Drawings ដូចរូប ។ នោះនឹងចេញផ្ទាំងដូចខាងក្រោម :

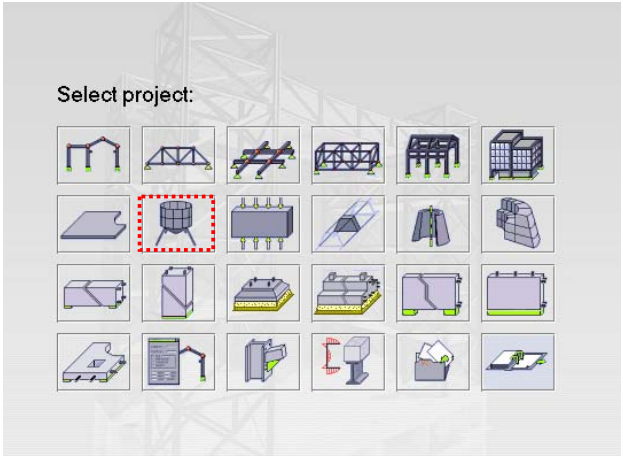


មេរៀនទី៥:

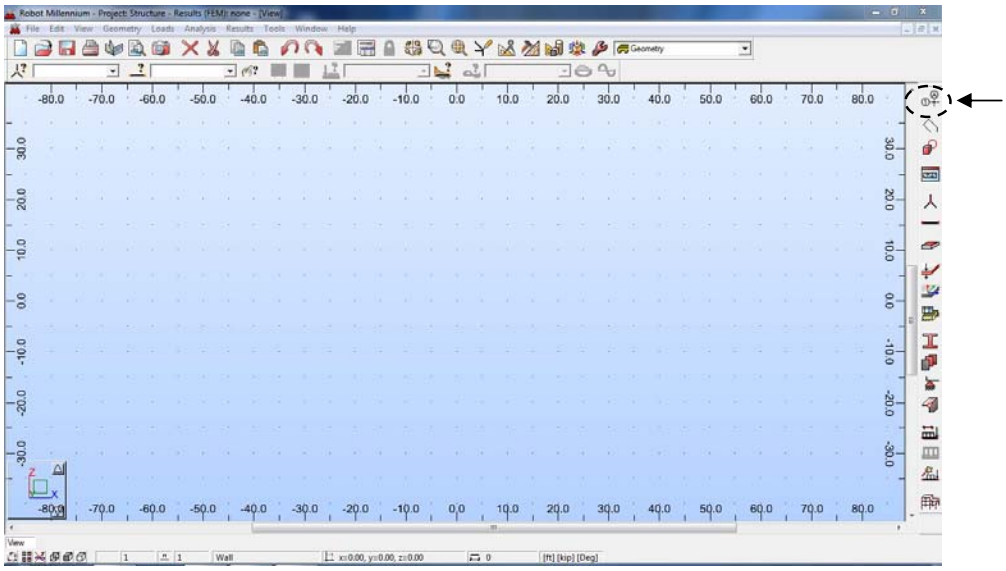
ការគណនារក្ខុចប្លង់ 3D

ជាមួយនឹងកម្មវិធី Autodest Robot Structural Analysis Prefessional2010(ARSAP)

នៅពេលដែលយើងបើកកម្មវិធី ជាដំបូងយើងនឹងបានឃើញ Screen ដូចខាងក្រោមនេះ បន្ទាប់ មកចុចជ្រើសរើសយក Icon  ដូចដែលបានបង្ហាញនៅក្នុងរូបខាងក្រោមនេះ



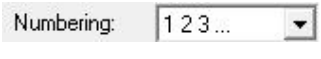
បន្ទាប់មកវានឹងបង្ហាញមកនូវ Screen ដូចខាងក្រោមនេះ



១- ការកំណត់យកអ័ក្ស Axis សំរាប់ការគូរ

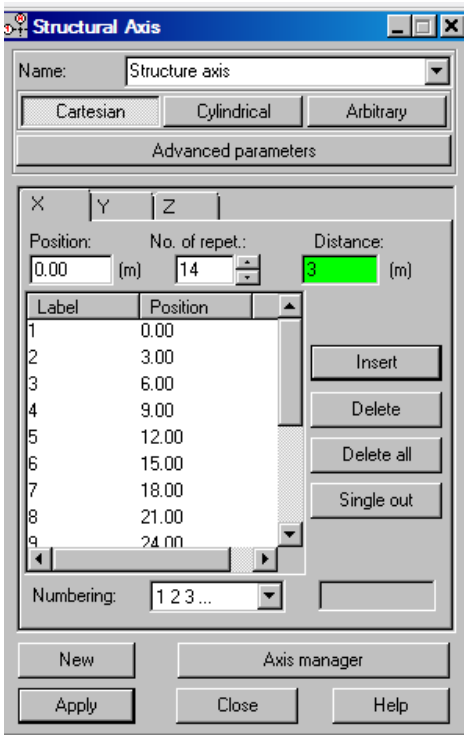
- ដំបូងយើងចុចលើ Icon  នេះដែលស្ថិតនៅខាងស្តាំនៃ Screen ដែលដូចបានបង្ហាញខាងលើ

១.១- ធ្វើការកំណត់លើអ័ក្សអាប់ស៊ីស X



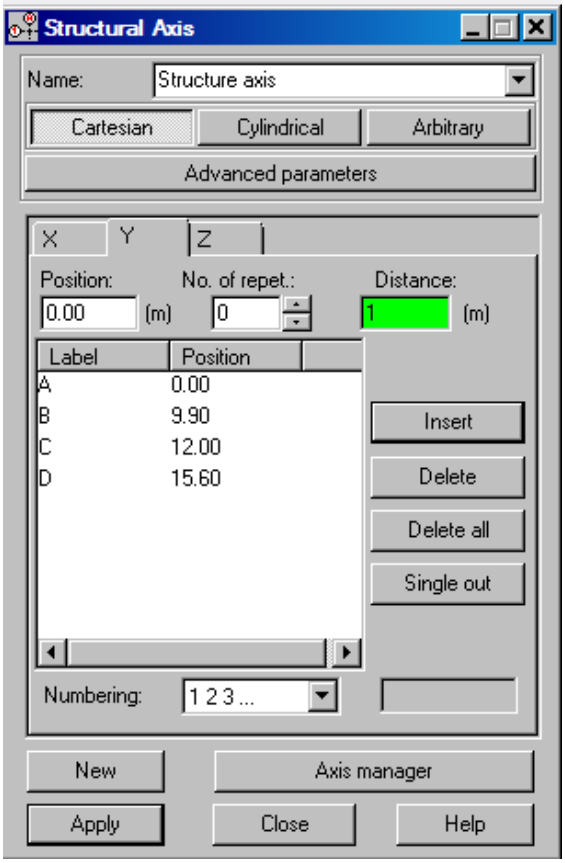
ដើម្បីងាយស្រួលក្នុងការគូរ និងងាយចំណាំ

- ត្រង់ប្រអប់ Position: វាយលេខ 0
- ត្រង់ប្រអប់ No of repet : វាយលេខ 14
- ត្រង់ប្រអប់ Distance : វាយលេខ 3.0 m
- បន្ទាប់សូមចុចលើ Button Insert



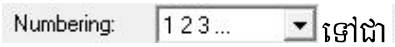
១.២- ការកំណត់អ័ក្សតាមទិសអរដោនេ

- ជាដំបូងសូមចុចលើ Tab Y បន្ទាប់មកសូមធ្វើការប្តូរនៅត្រង់ប្រអប់ Numbering គឺប្តូរពី លេខ ទៅ អក្សរ
- ត្រង់ប្រអប់ Position : វាយលេខ 0
- ត្រង់ប្រអប់ No. of repet: វាយលេខ 1
- ត្រង់ប្រអប់ Distance : វាយលេខ 9.9 m
- រួចចុចលើ Button Insert
- Position:9.9 m
- បន្ទាប់មកត្រង់ប្រអប់ No. of repet វាយលេខ 1
- ហើយត្រង់ប្រអប់ Distance : 2.1 m
- រួចចុចលើ Button Insert
- Position: 12.0 m
- No. of repet: 1
- Distance: 3.6 m



១.៣- ការកំណត់លើអ័ក្សរយៈកំពស់ Z

- ជាដំបូងសូមចុចលើ Tab Z រួចសូមធ្វើការកែត្រង់ប្រអប់ Numbering ដោយប្តូរពី



ទៅជា


ដើម្បីធ្វើអោយកាន់តែងាយចងចាំ

- ត្រង់ប្រអប់Position : វាយលេខ -1.6
- ត្រង់ប្រអប់ No.of repet: វាយលេខ 0
- ត្រង់ប្រអប់ Numbering: Value = Level:-1.6 m
- រួចចុចលើ Button 

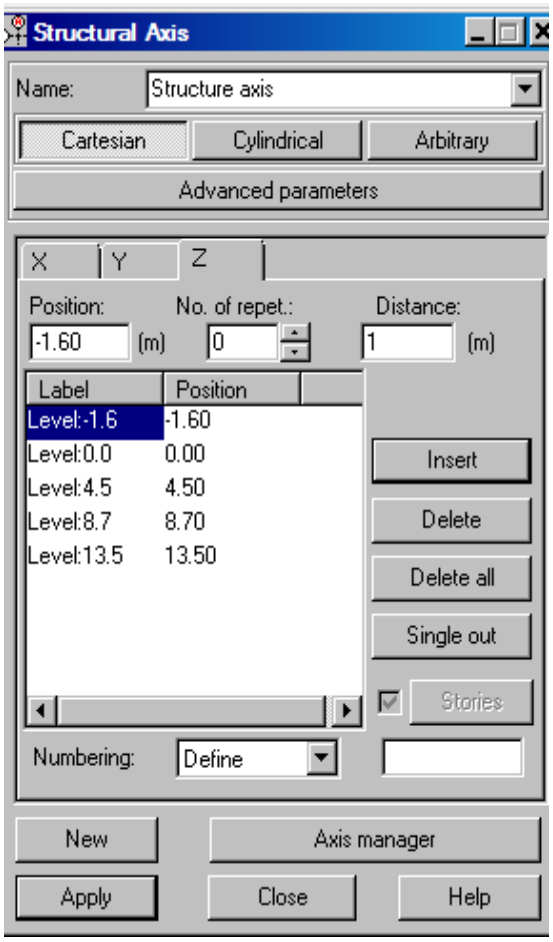
- ត្រង់ប្រអប់ វាយលេខ 3.25m
- Position:0.0
- No. of repet: 1
- Distance: 0
- Numbering : Value = Level:0.0
- រួចចុចលើ Button 

- Position: 4.5 m
- Numbering: Value = Level:4.5
- No. of repet: 1
- Distance: 0
- រួចចុចលើ Button 

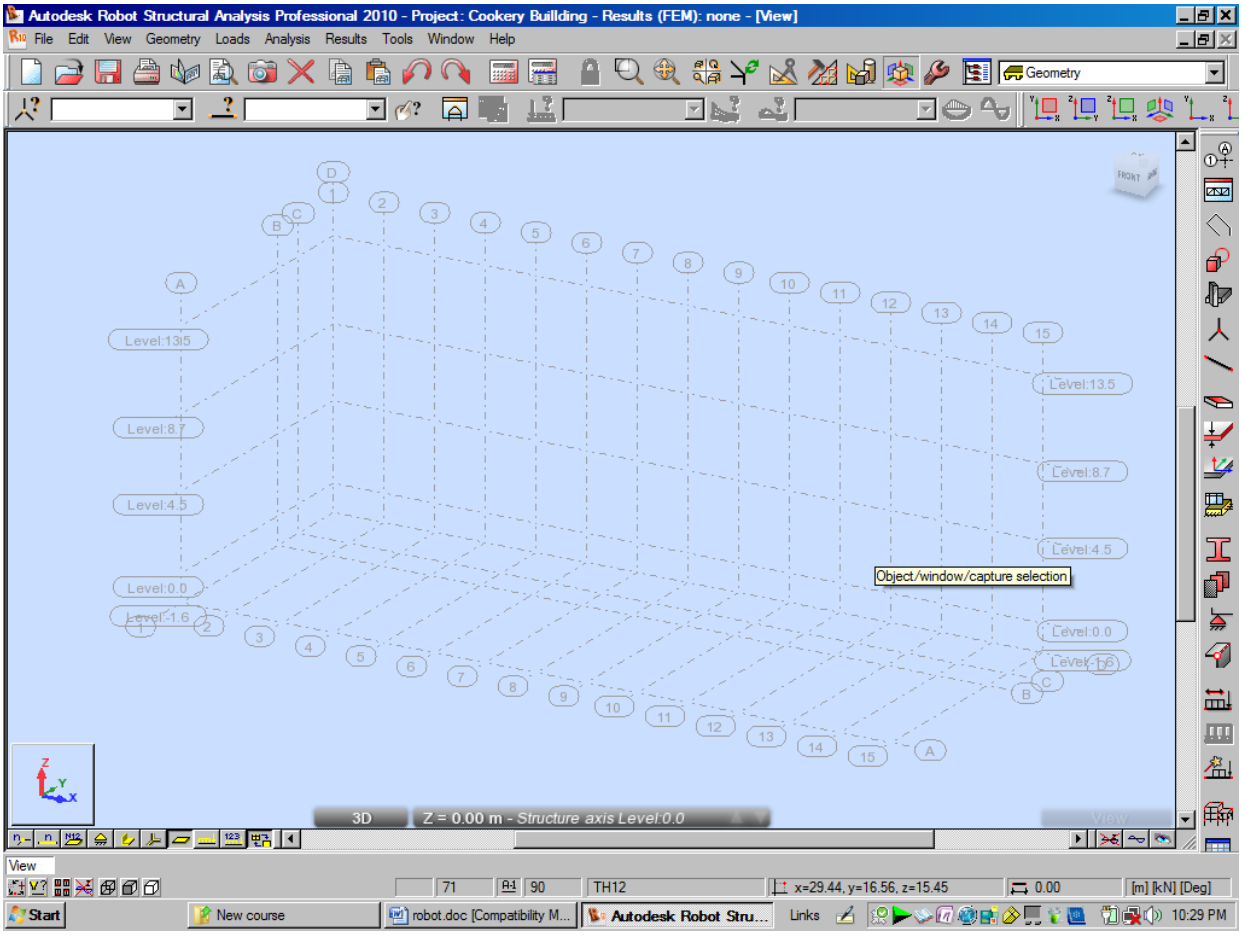
- Position: 8.7 m
- Numbering: Value = Level: 8.7
- No. of repet: 1
- Distance: 0
- រួចចុចលើ Button 

- Position: 13.5 m
- Numbering: Value = Level:13.5
- No. of repet: 1
- Distance: 0
- រួចចុចលើ Button 

- បន្ទាប់មកចុច Button Apply > View > 3D Project XYZ
-



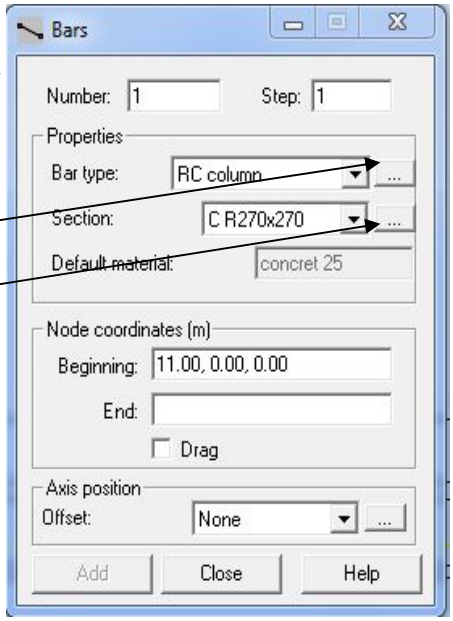
ក្រោយពីយើងបានកំណត់រាល់អ័ក្សទាំងអស់ហើយ វានឹងបង្ហាញចេញមកនូវផ្ទាំងដូចខាងក្រោមនេះ



រូបបង្ហាញជាលក្ខណៈ 3D Project XYZ

២- ការគូរធ្នឹម និងសសរ

- ជំហ៊ានដំបូងយើងត្រូវធ្វើការបង្កើតមុខកាត់ធ្នឹម រឺសសរដែលយើងចង់បានជាមុនសិន
- ដើម្បីធ្វើការបង្កើតមុខកាត់យើងត្រូវ
- ដំបូងសូមចុចលើ Icon ———— នេះសំរាប់កំណត់ប្រភេទរបារជាធ្នឹម រឺជាសសរ



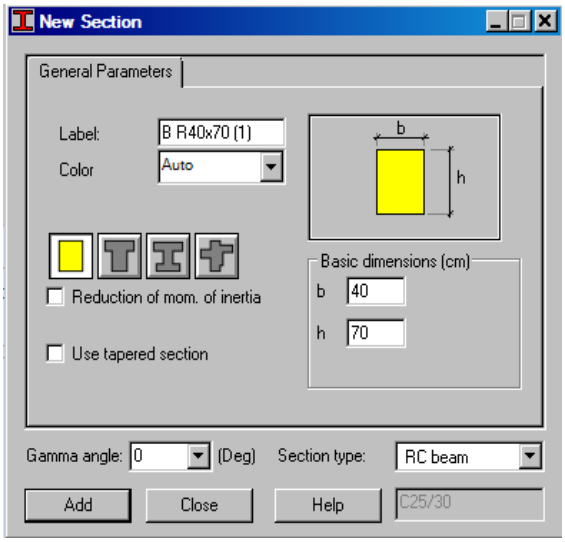
សំរាប់កំណត់ប្រភេទរបារ ធ្នឹម រឺ សសរ

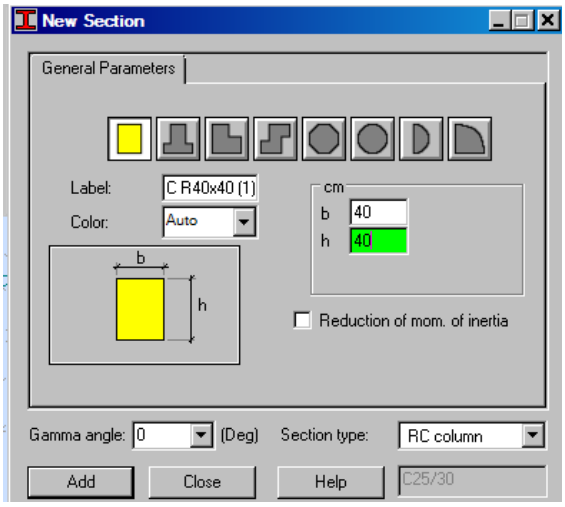
សំរាប់បង្កើតមុខកាត់ ធ្នឹម រឺ សសរ

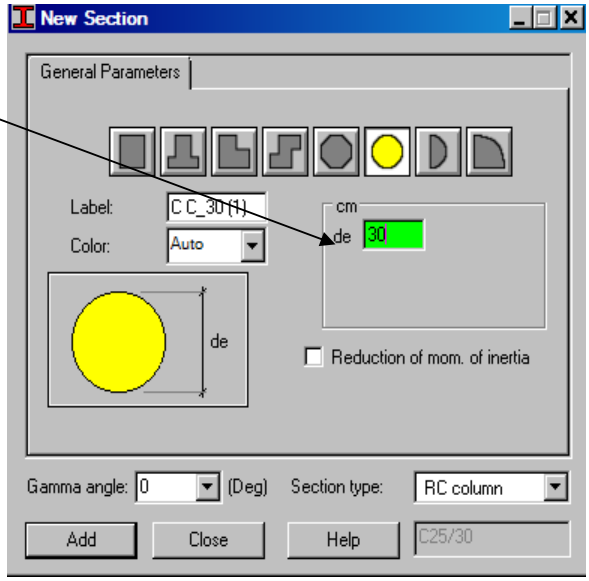
២.១- ការបង្កើតមុខកាត់ធ្នឹម

- Section Type: RC Beam
- Basic dimensions (cm)
  - b = 40, h = 60 ចុច Command button Add
  - b = 30, h = 60 ចុច Command button Add
  - b = 40, h = 70 ចុច Command button Add
  - b = 30, h = 70 ចុច Command button Add
  - b = 30, h = 50 ចុច Command button Add
  - b = 30, h = 55 ចុច Command button Add

- Section Type: RC Column
- Basic Dimensions(cm)
  - b = 40, h = 60 ចុច Command button Add
  - b = 30, h = 60 ចុច Command button Add
  - b = 40, h = 70 ចុច Command button Add
  - b = 30, h = 70 ចុច Command button Add
  - b = 30, h = 50 ចុច Command button Add
  - b = 30, h = 55 ចុច Command button Add
  - de = 30 cm (D ជាអង្កត់ផ្ចិតសសរ ដែលមានរាមូល)

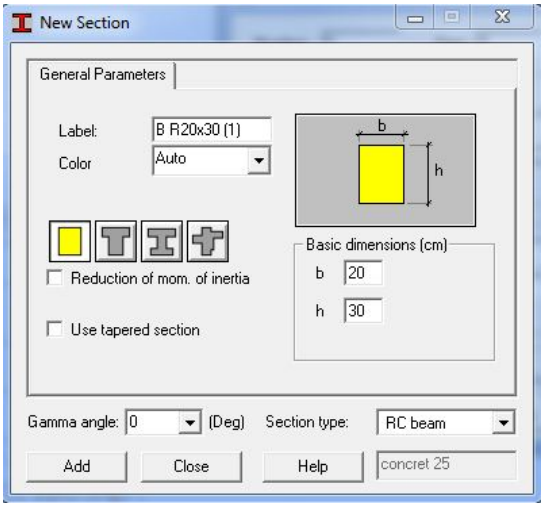






- នៅក្នុងប្រអប់ Bar type: ចុចយក RC beam
- នៅក្នុង Section: ចុចលើរូប Icon ... ដើម្បីបង្កើតមុខកាត់ធ្នឹម 20 x 30

- នៅត្រង់ Basic dimensions : វាយតំលៃក្នុងប្រអប់ b = 20cm និង h = 30cm

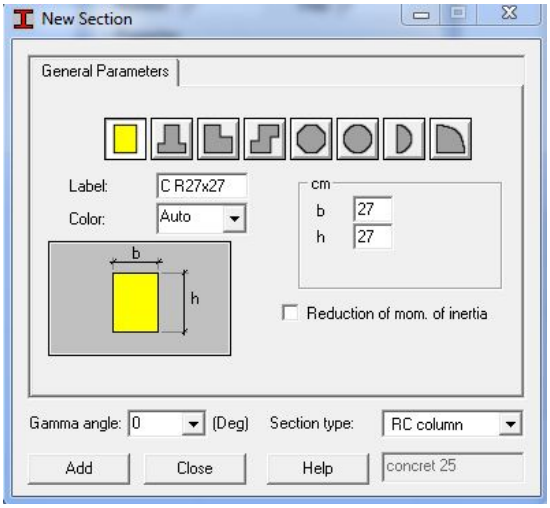


- បន្ទាប់មកសូមចុចលើ Button Add

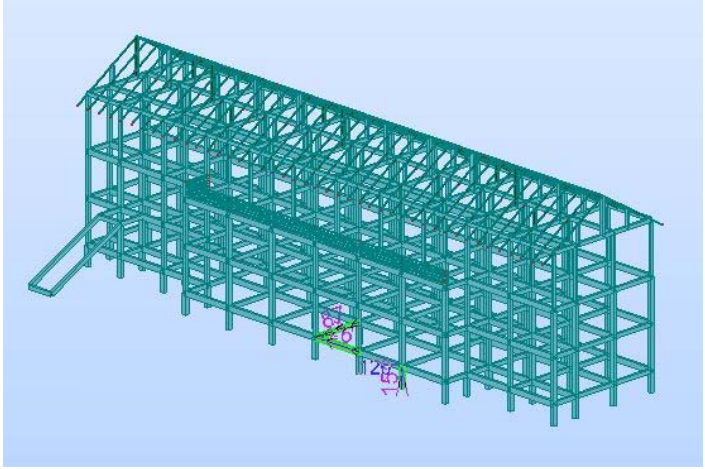
- បន្ទាប់សូមបង្កើតមុខកាត់ជាបន្តបន្ទាប់ដូចដែលយើងចង់បង្កើត ហើយសំរាប់ការបង្កើតគឺដូចគ្នាទៅនឹងការបង្កើតមុខកាត់ 20cm x 30cm ដូចគ្នាដែរ ។

២.២- ការបង្កើតមុខកាត់សសរ 2.7cm x 2.7cm

- នៅត្រង់ Basic dimensions: វាយតំលៃលេខក្នុងប្រអប់ b=27cm និង h=27cm
- បន្ទាប់សូមចុចលើ Button Add



បន្ទាប់ពីយើងធ្វើការគូររាល់គ្រប់ Structure អស់ហើយ យើងនឹងទទួលបាន នូវរូបដូចបង្ហាញខាងក្រោមនេះ



២.៣- ការកំណត់បន្ទុកដែលមានអំពើមកលើធ្មឹម

២.៣.១- កំណត់ឈ្មោះបន្ទុក (Load Cases)

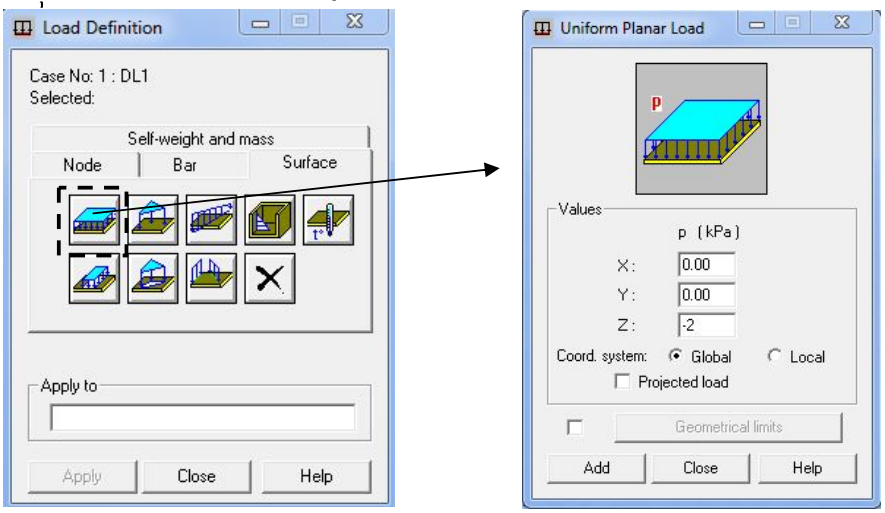
- ចុចលើ Icon  នេះដែលស្ថិតនៅខាងស្តាំនៃអេក្រង់
- ដំបូងយើងត្រូវសរសេរឈ្មោះ Self-weight នៅក្នុងប្រអប់ Name
- បន្ទាប់មកចុច New
- នៅត្រង់ Nature រើសយក live ហើយវាយឈ្មោះ LL
- រួចចុចលើ New
- ត្រង់ Nature រើសយក lead ហើយវាយឈ្មោះ DL
- ត្រង់ Nature រើសយក lead ហើយវាយឈ្មោះ Wall
- រួចចុច New → ចុងក្រោយចុច Close ដើម្បីបិទចោល

៣.១ បន្ទុកដែលត្រូវកំណត់

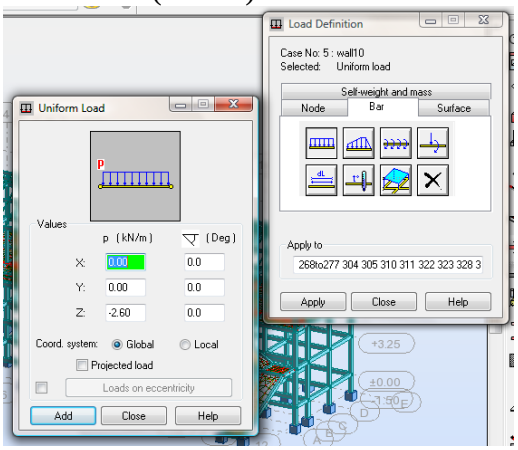
-ក្រោយពីការប្រមូលបន្ទុកហើយយើងទទួលបានតំលៃបន្ទុកដូចខាងក្រោម:

-DL1=Seft-Load
-DL2=1.2KN/m^2
-LL1=2 KN/m^2
-LL2=2 KN/m^2
-Wall10=2.6 KN/m
-Wall20=14.92 KN/m
-Roof DL=35Kg/m, Roof LL=50Kg/m
-Wind Load=35Kg/m

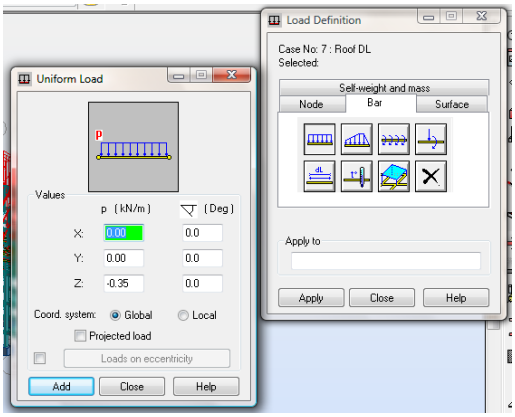
-ការដាក់បន្ទុក DL2, LL1, LL2 ត្រូវដាក់លើ Slab



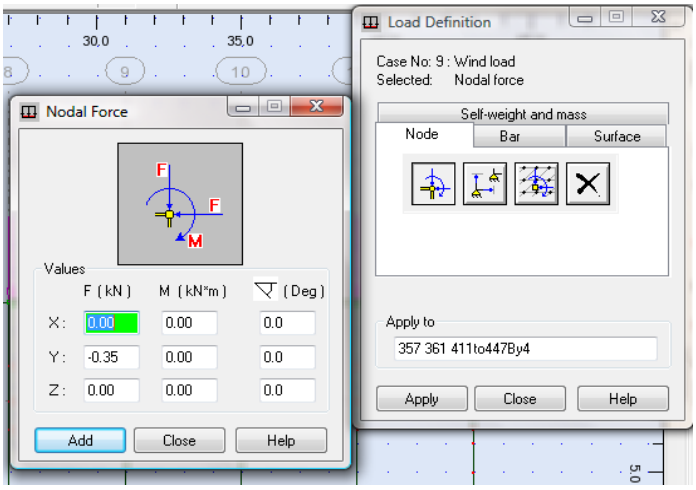
- ការដាក់បន្ទុក Wall10, Wall20 ត្រូវដាក់លើ Beam



- ការដាក់បន្ទុក Roof DL, Roof LL ត្រូវដាក់លើដែរនែង



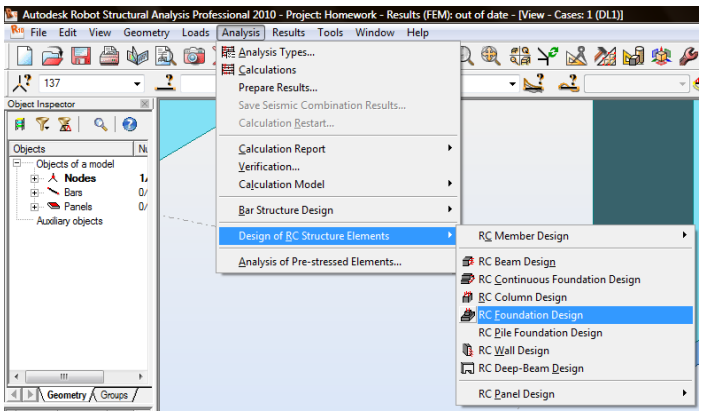
- ការដាក់បន្ទុក Wind Load



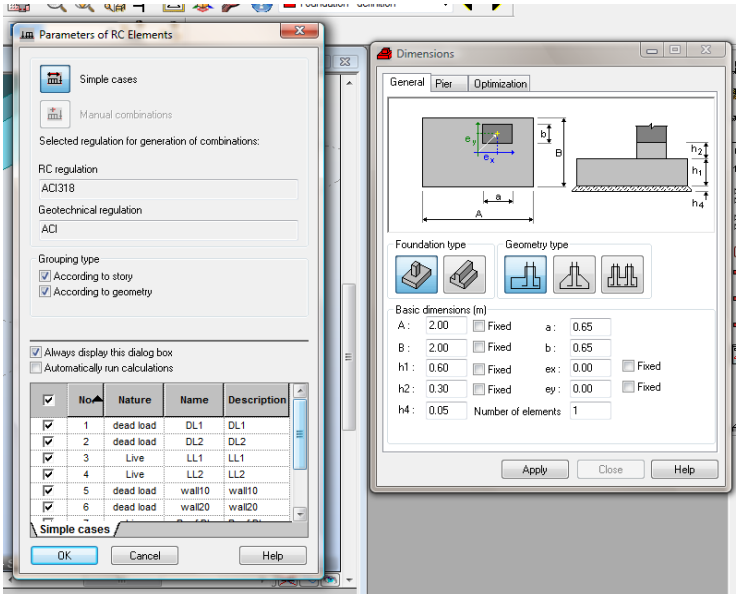
-បន្ទាប់ពីដាក់បន្ទុកហើយត្រូវចុច  ដើម្បីគណនាកំលាំងក្នុង ម៉ូម៉ង់ កំលាំងកាត់ទទឹង ។

៤.១ ការគណនាគ្រឹះ ត្រូវSelect Foot

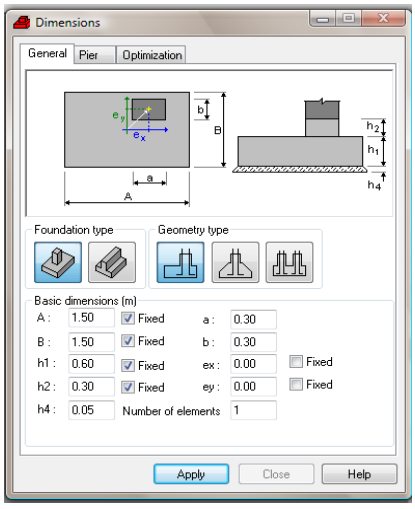
បន្ទាប់មកយើងត្រូវចូលកាន់ ដូចរូបដែលបានបង្ហាញខាងក្រោមនេះ ដើម្បីយើងទាញយកគ្រឿងបង្គុំណាមួយមកធ្វើការ គណនា



ចូលបន្តបំមកយើងបានរូបដូចខាងក្រោម:

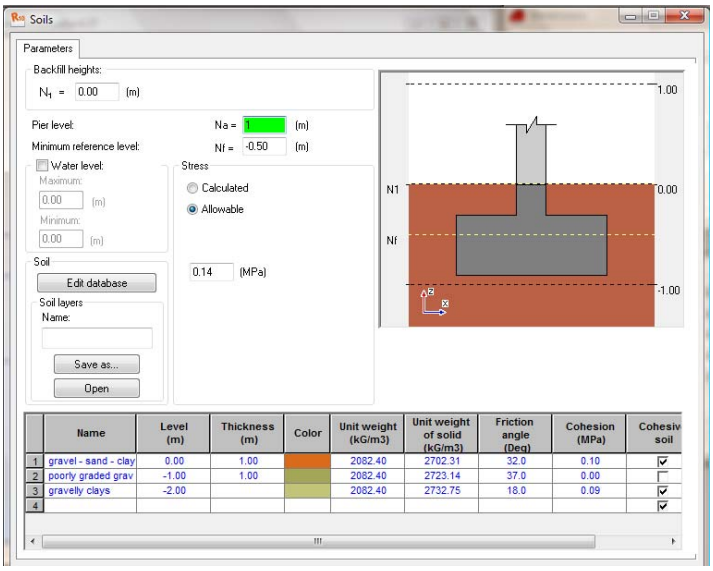


យើងត្រូវកំណត់គ្រឹះជើងតាង:

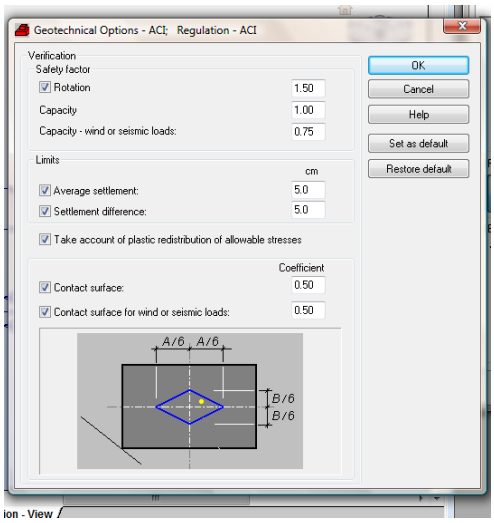


រួចចុចពាក្យ Apply

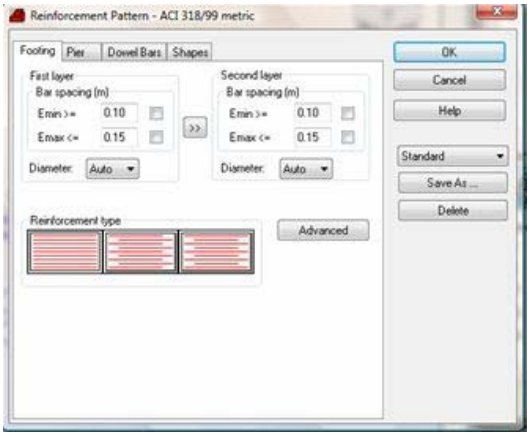
-រួចកំណត់ស្រទាប់ដីនៅខាងក្រោមគ្រឹះ



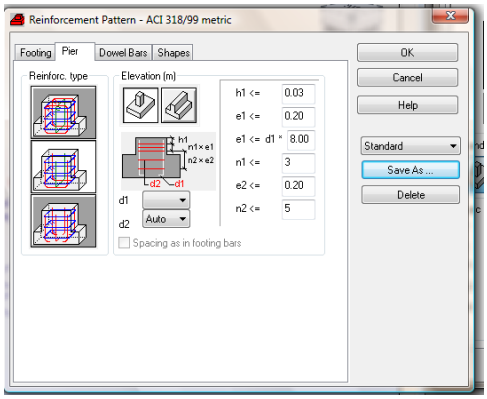
	Name	Level (m)	Thickness (m)	Color	Unit weight (kG/m3)	Unit weight of solid (kG/m3)	Friction angle (Deg)	Cohesion (MPa)	Cohesiv soil
1	gravel - sand - clay	0.00	1.00		2082.40	2702.31	32.0	0.10	☑
2	poorly graded grav	-1.00	1.00		2082.40	2723.14	37.0	0.00	☐
3	gravelly clays	-2.00			2082.40	2732.75	18.0	0.09	☑
4									☑



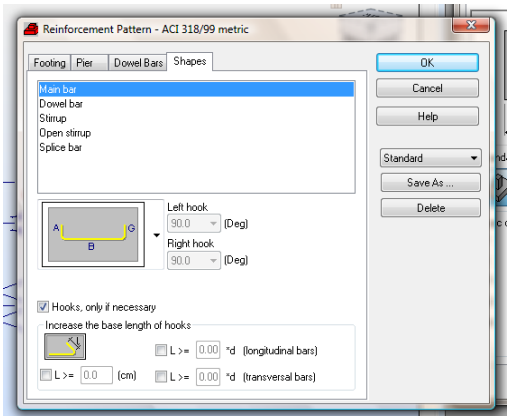
-ការកំណត់ Reinforcement Pattern



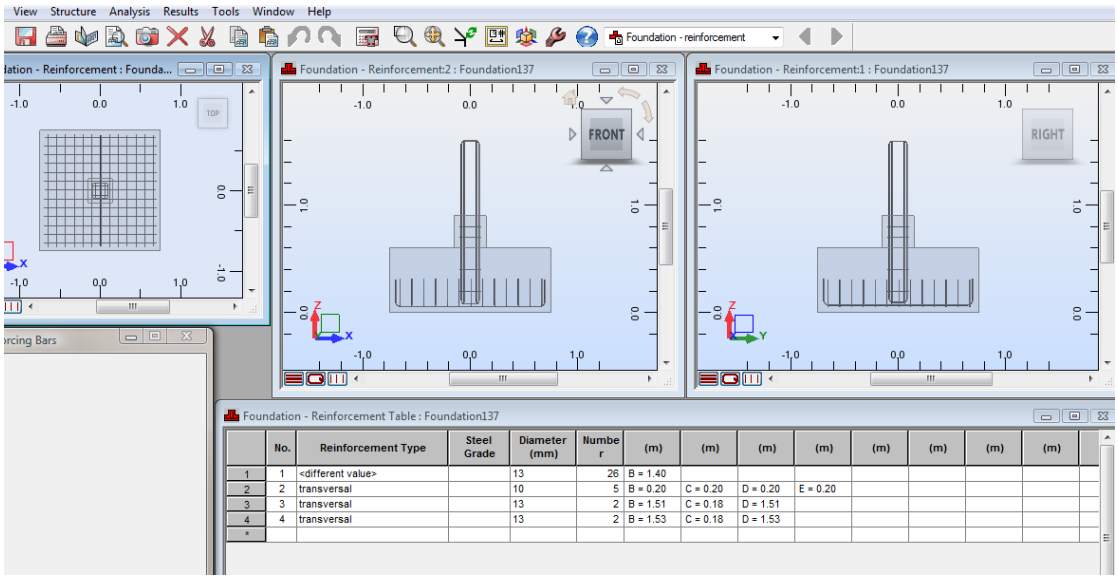
-ការកំណត់របស់ជើងតាង



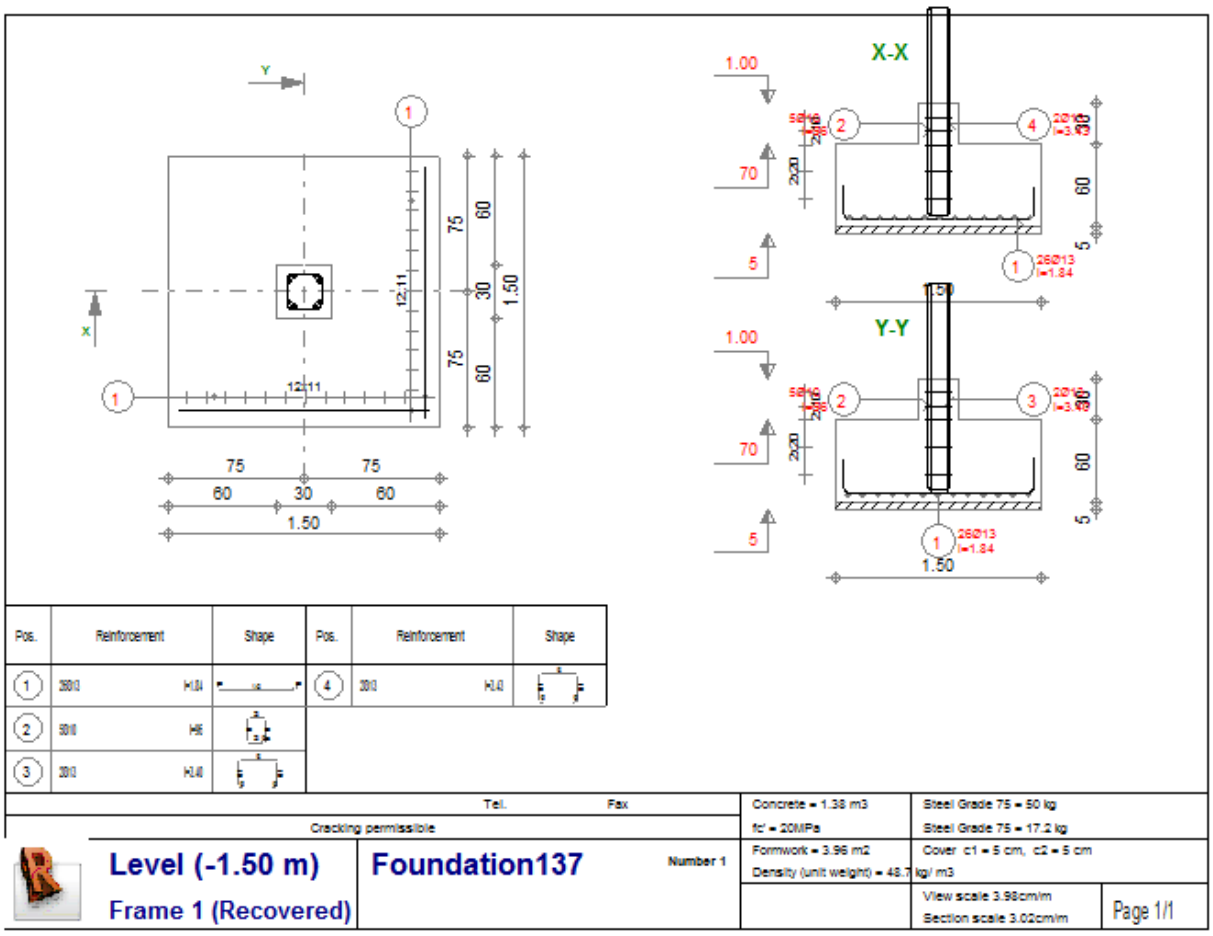
-ការកំណត់រាងរបស់ដែក



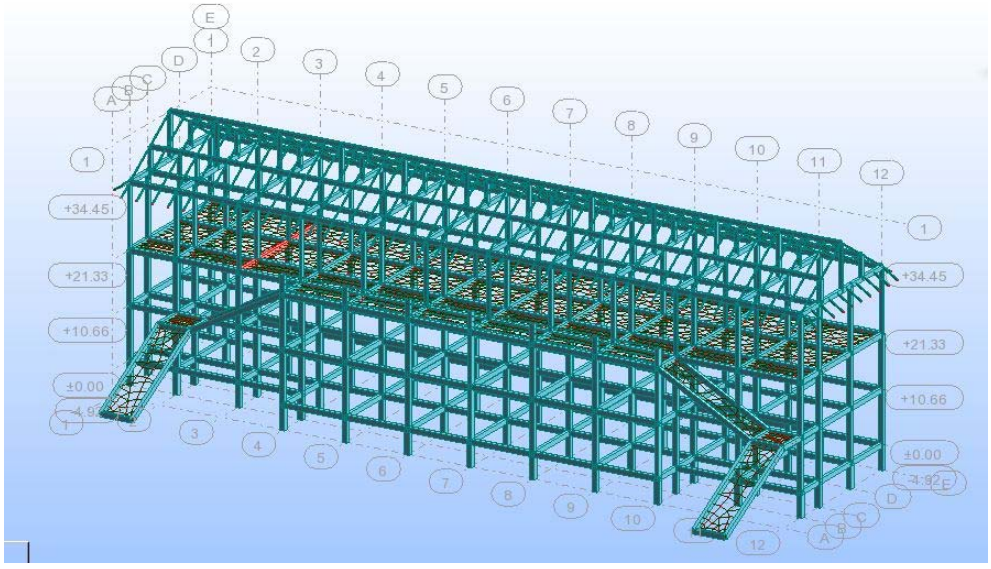
-ចាប់ផ្តើមគណនារកសរសៃដែក Start Calculation



ការបង្ហាញ Drawing

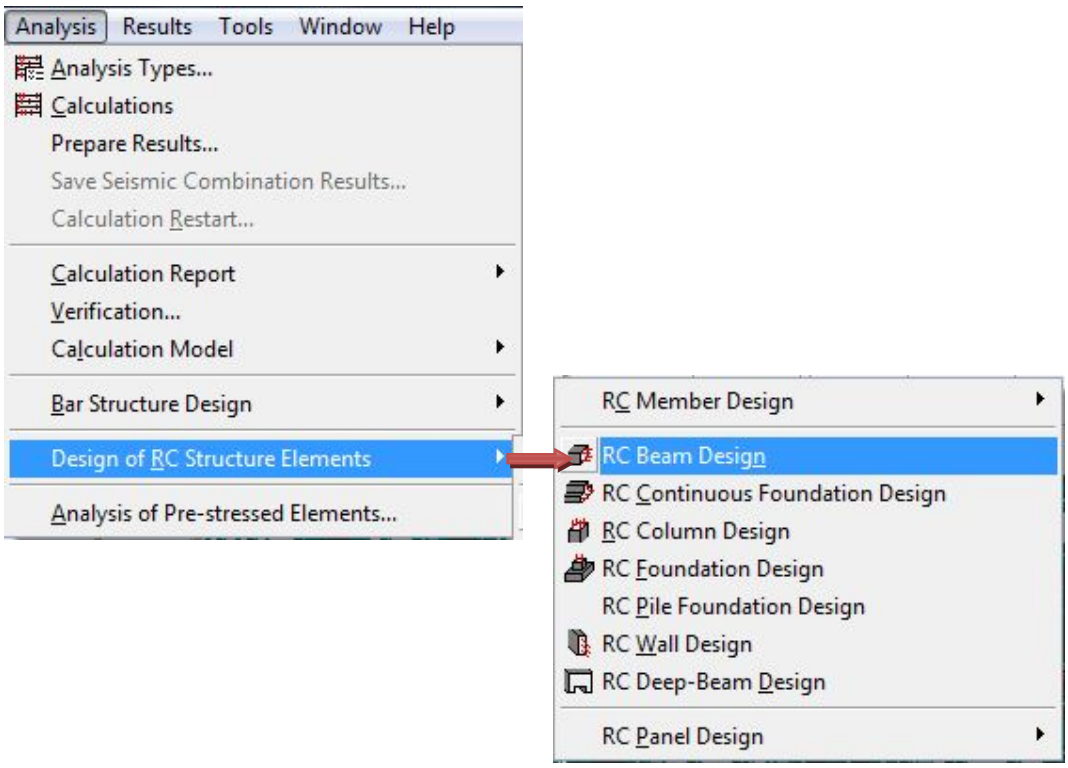


៤.១ ការគណនាធ្នឹមមកធ្វើការសិក្សា

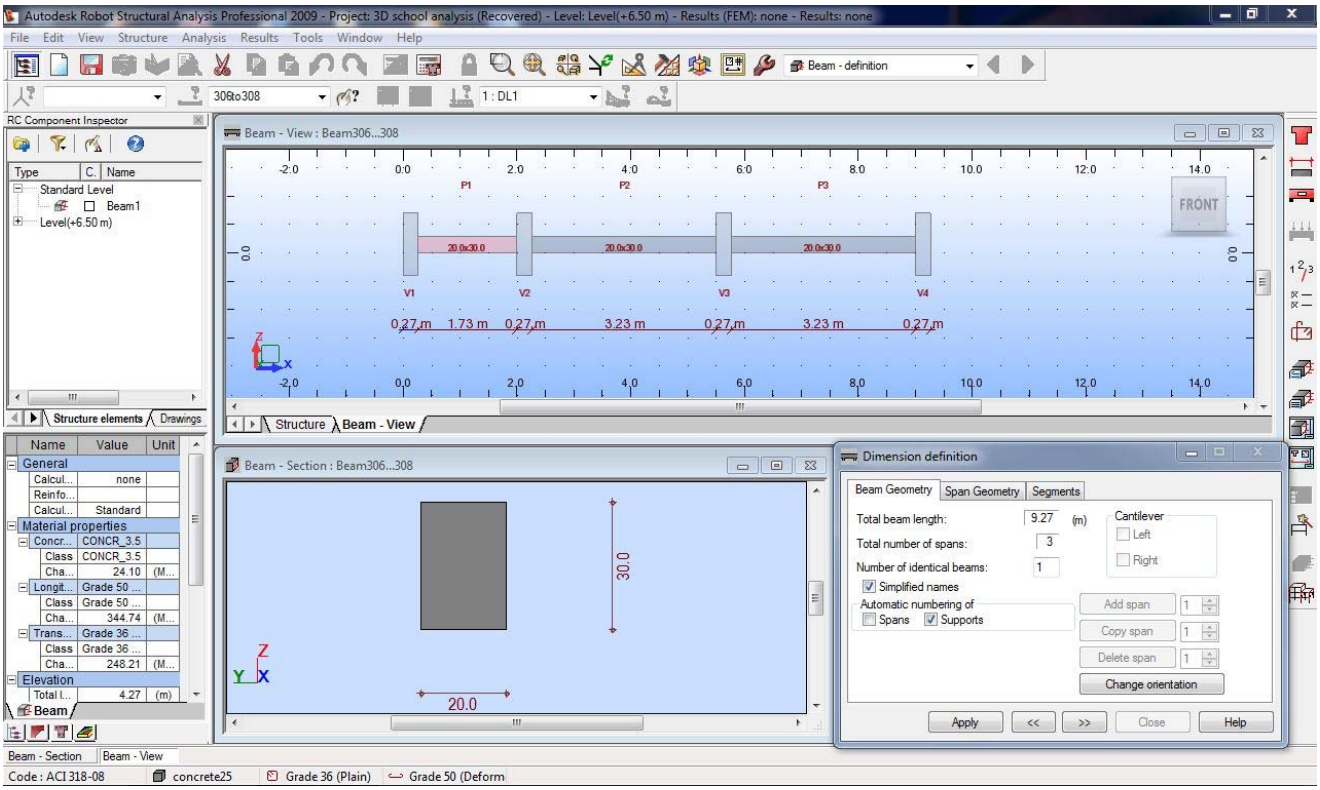


ដើម្បីធ្វើការសិក្សាលើធ្នឹមនេះបានយើងត្រូវអនុវត្តន៍តាមវិធីដូចខាងក្រោមនេះ :

Analysis → Design of RC Structure Elements → RC Beam Design

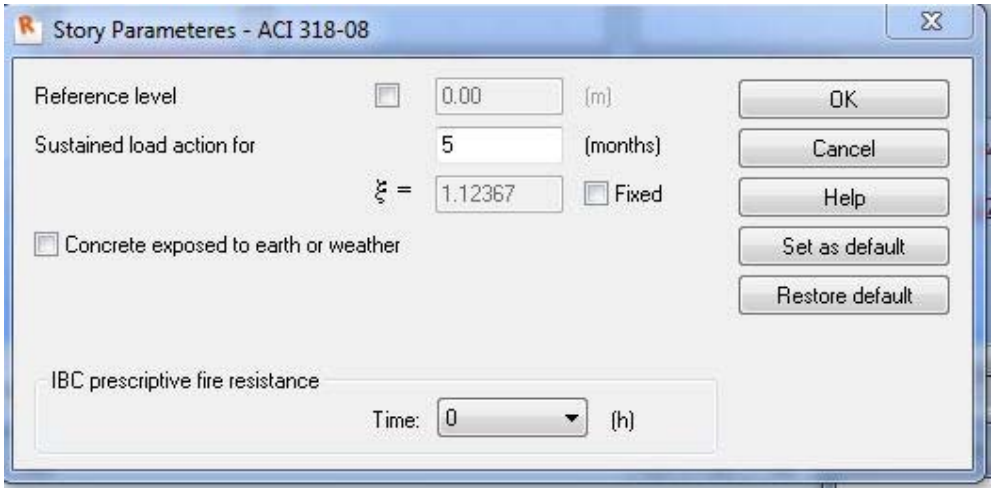


បន្ទាប់ពីយើងធ្វើការជ្រើសរើសដូចរូបខាងលើមក វានឹងបង្ហាញដូច Screen ខាងក្រោមនេះ



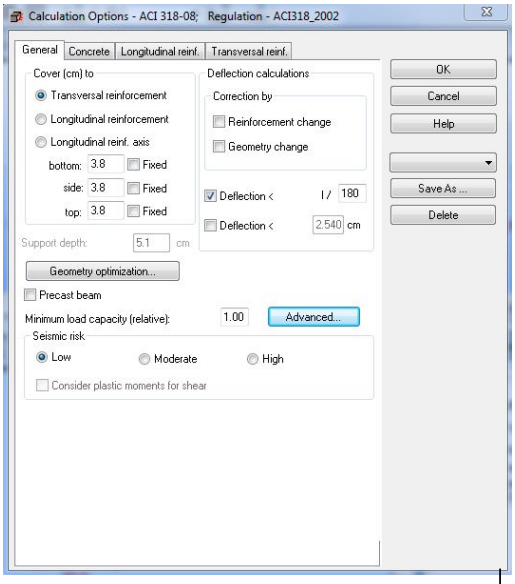
បន្ទាប់មកយើងត្រូវធ្វើការកំណត់នូវ Parameter មុននឹងចាប់ផ្តើមធ្វើការគណនា និង Analysis

- ជាដំបូងយើងត្រូវធ្វើការកំណត់លើ Story Parameters ដោយចុចលើ Icon 

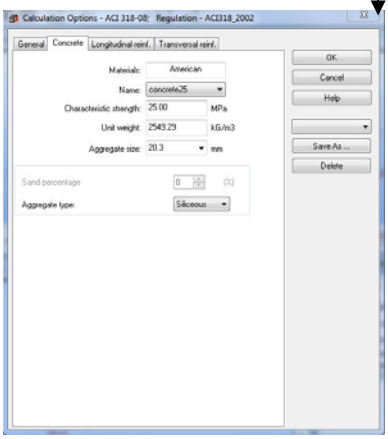


- បន្ទាប់មកយើងធ្វើការកំណត់លើ Calculation Option ដោយចុចលើ Icon 
- រួចមកយើងត្រូវធ្វើការកំណត់លើ Parameter ដូចខាងក្រោមនេះជាបន្តបន្ទាប់ :

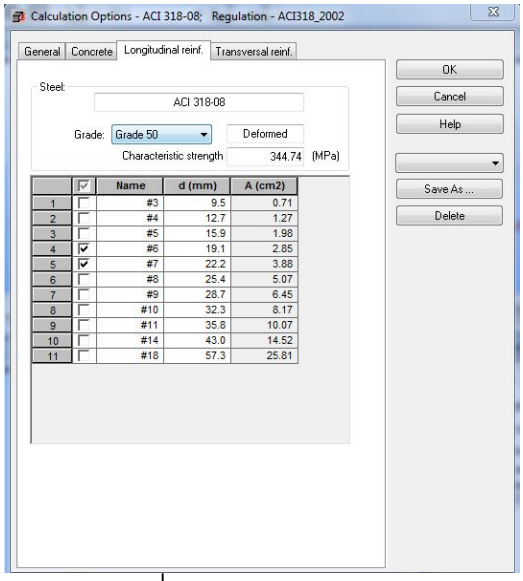
General →



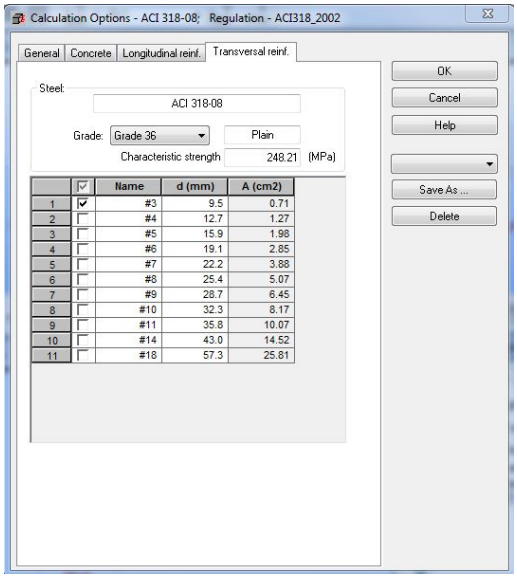
Concrete →



Longitudinal →



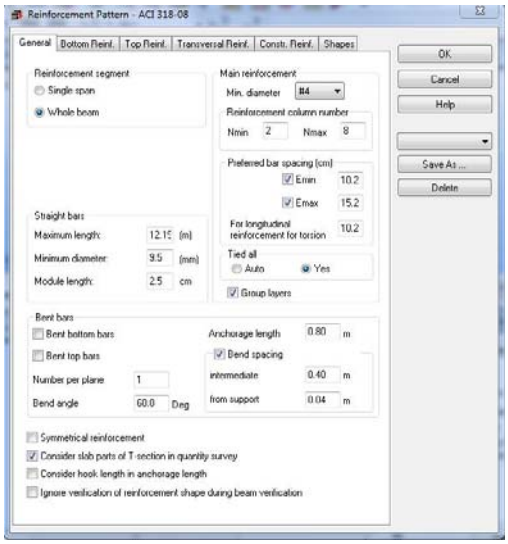
Transversal reinf →



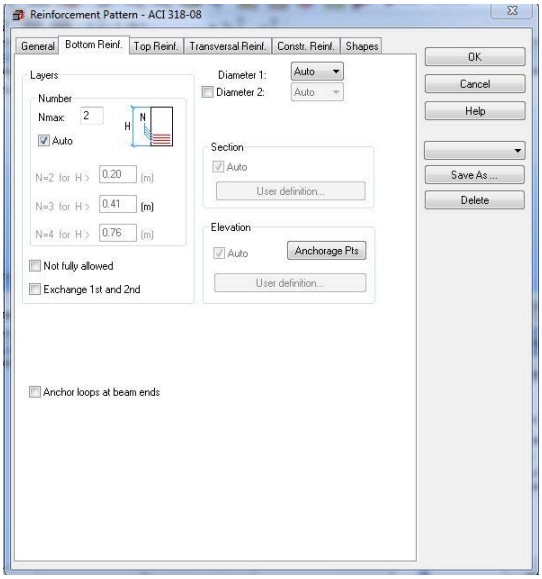
បន្ទាប់មកយើងចុចលើ Save as ដើម្បីធ្វើការកំណត់ឈ្មោះ

- បន្ទាប់មកយើងធ្វើការកំណត់លើ Parameter Reinforcement Pattern

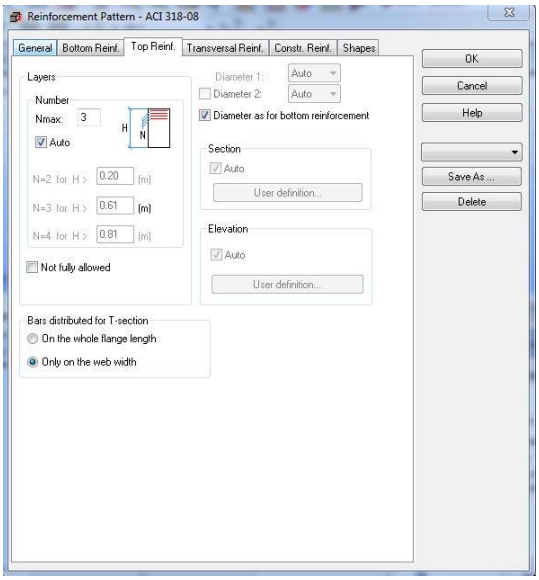
General →



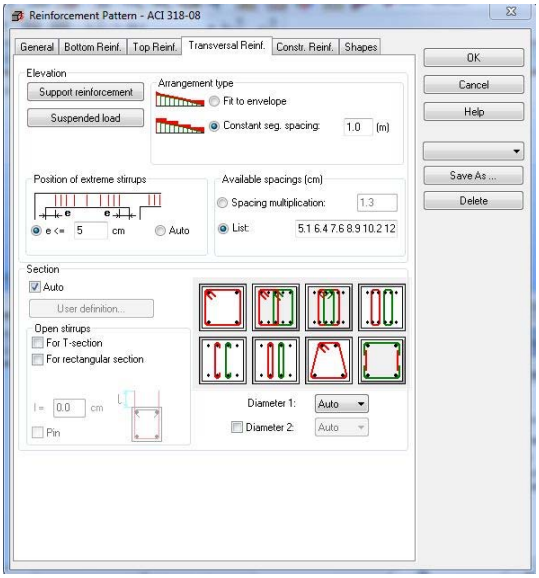
Bottom reinforcement →



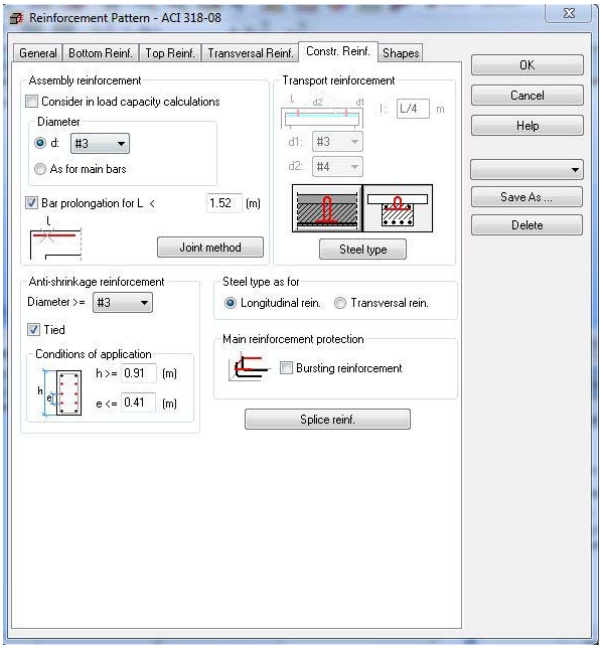
Top reinforcement →



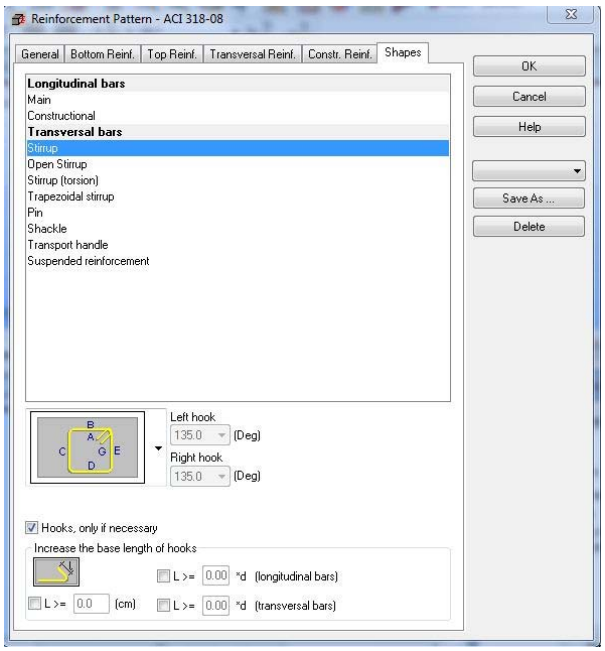
Transversal reinforcement →



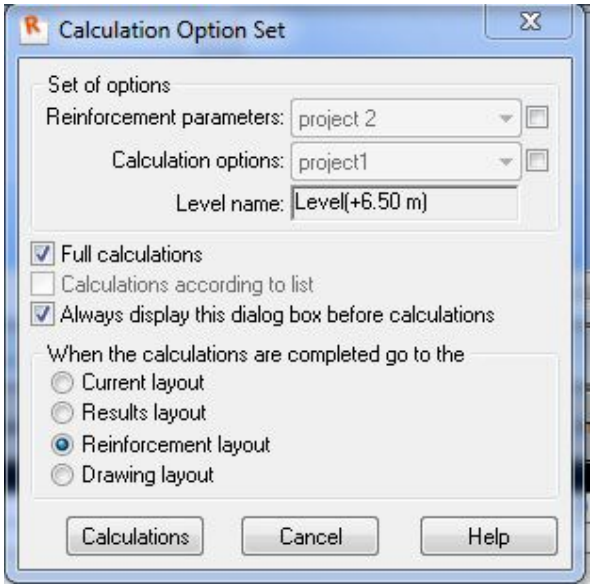
Construction reinforcement →



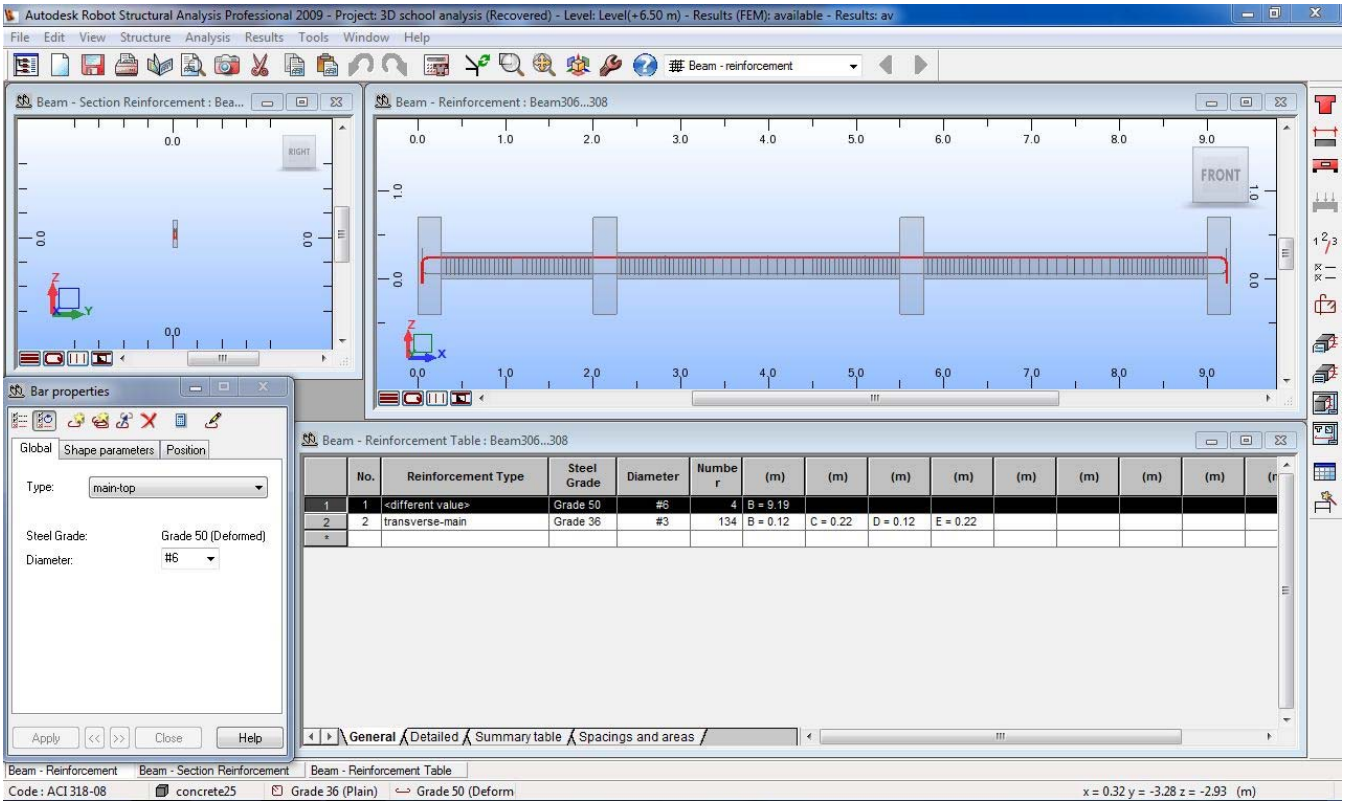
Shape →



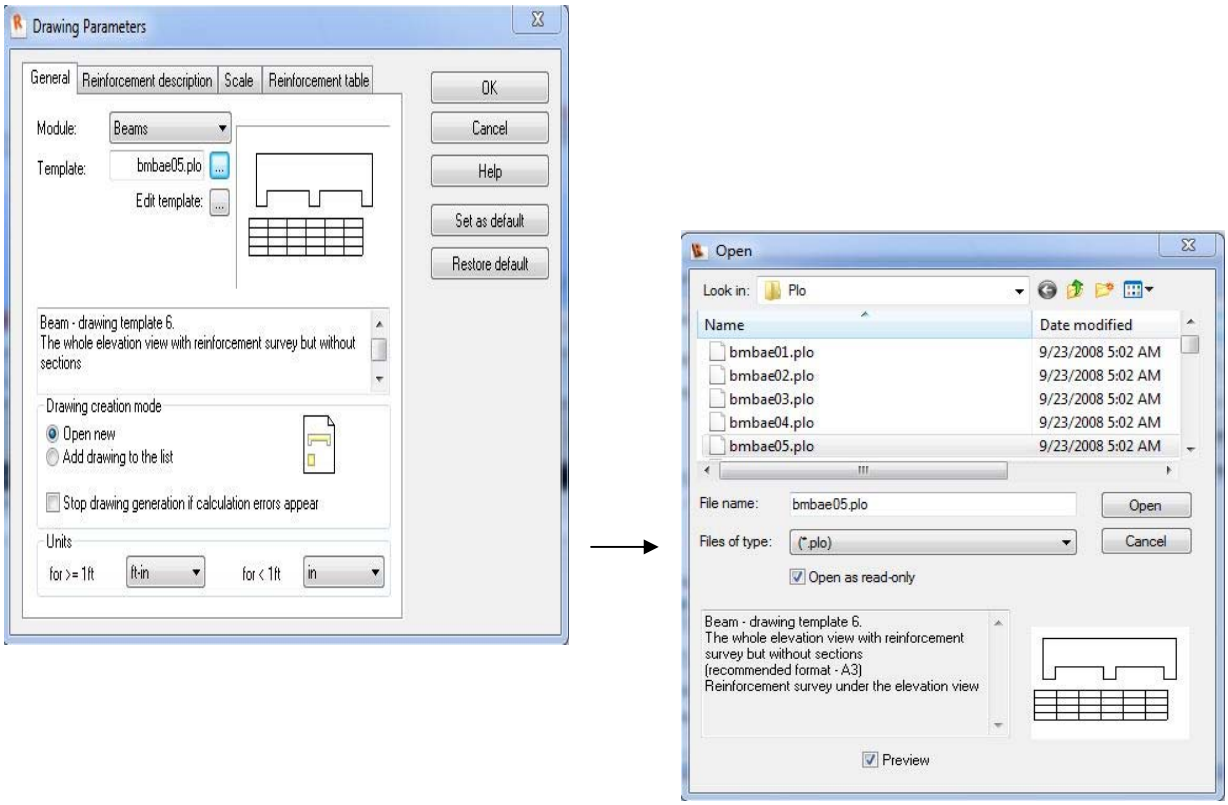
- បន្ទាប់ពីយើងធ្វើការកំណត់រួចរាល់នូវ Parameter ទាំងអស់នេះហើយយើងត្រូវ Save
- បន្ទាប់យើងធ្វើការគណនាដោយចុចលើ Icon 
- ពេលនោះវានឹងបង្ហាញដូច Screen ខាងក្រោមនេះ



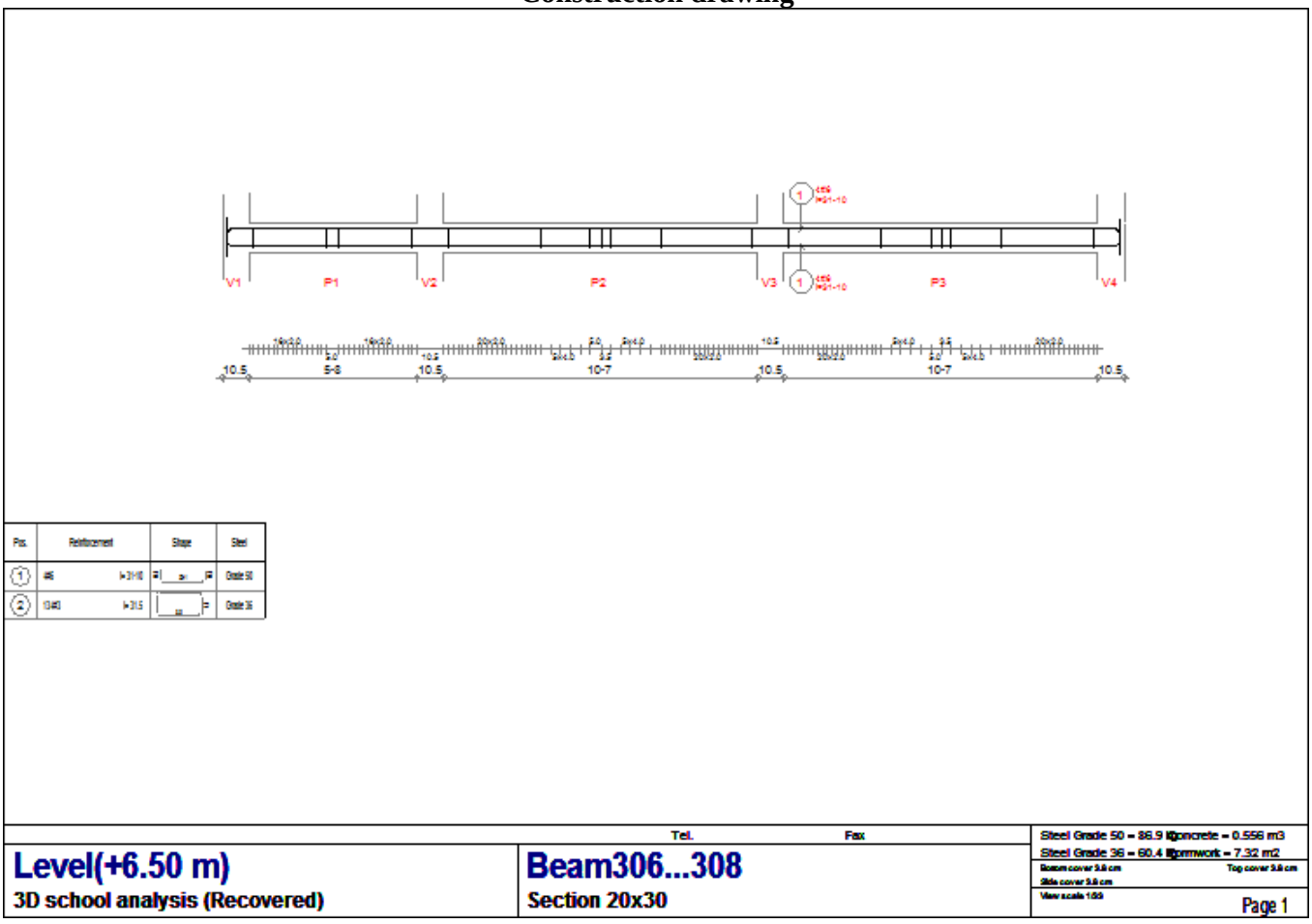
- បន្ទាប់មកចុចលើ Button Calculations 



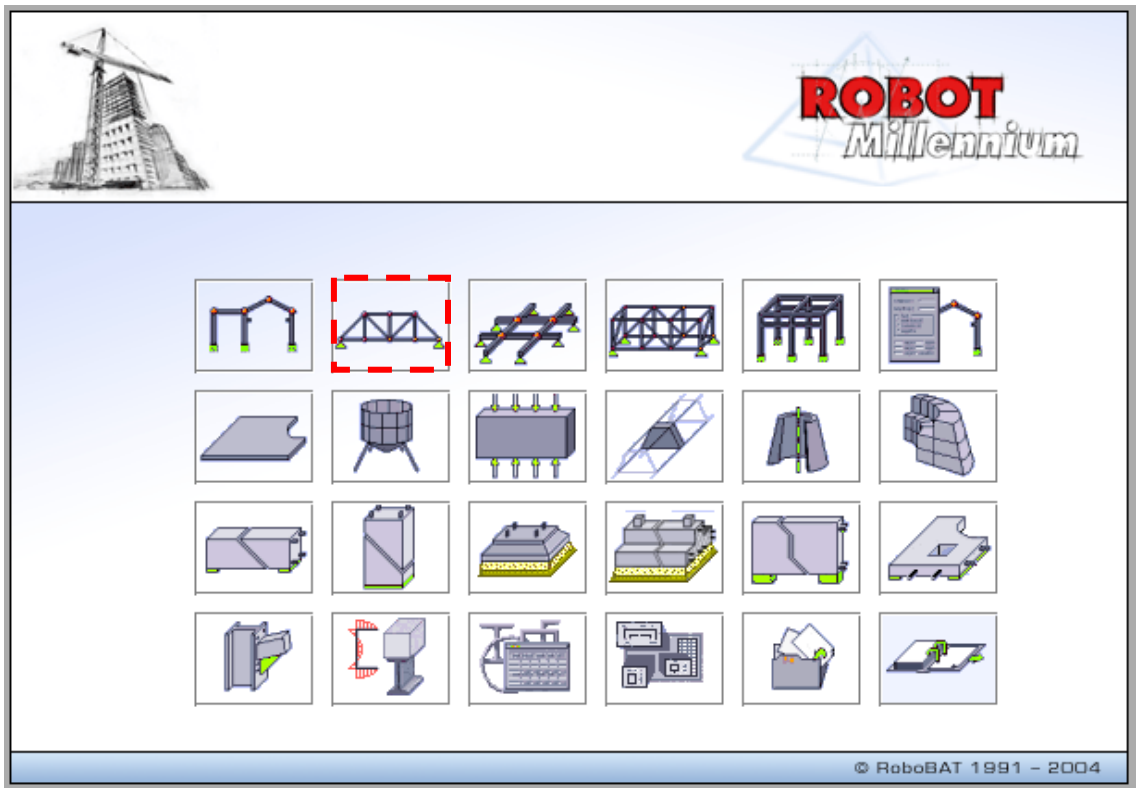
បន្ទាប់មកយើងធ្វើការកំណត់លើ Drawing option ដើម្បីបង្កើតប្លង់សំរាប់ការសាងសង់

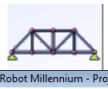


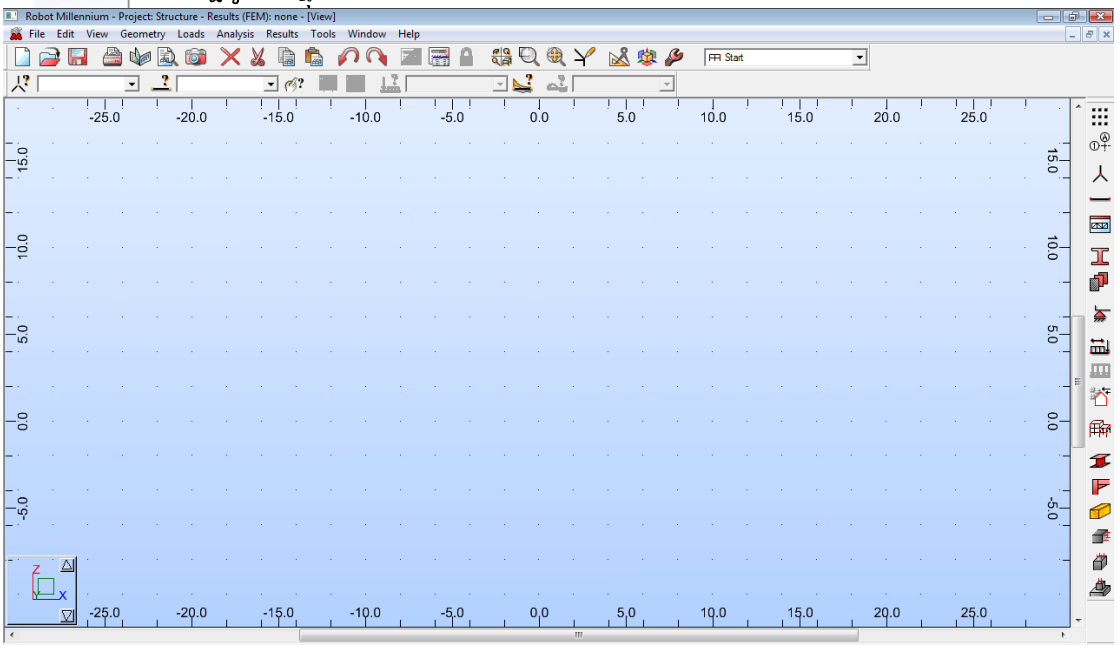
Construction drawing



មេរៀនទី៦: Truss 2D Design



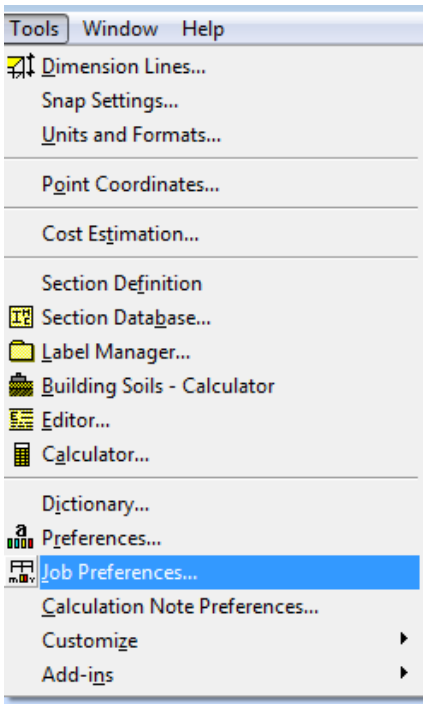
ចុចលើ Icon  ដើម្បីចូរទៅក្នុង 2D Truss



www.yerngshare.blogspot.com

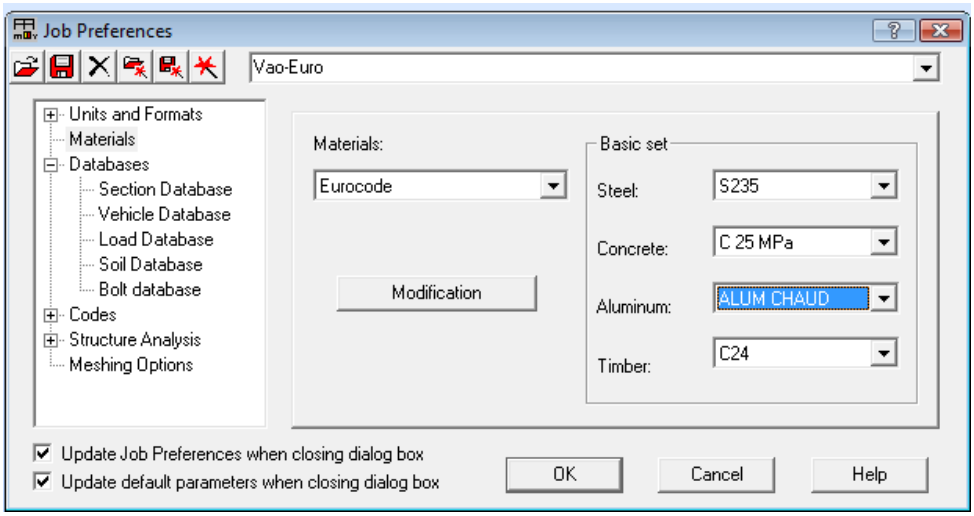
១. ការកំណត់ Materials និង Code សំរាប់គណនា:

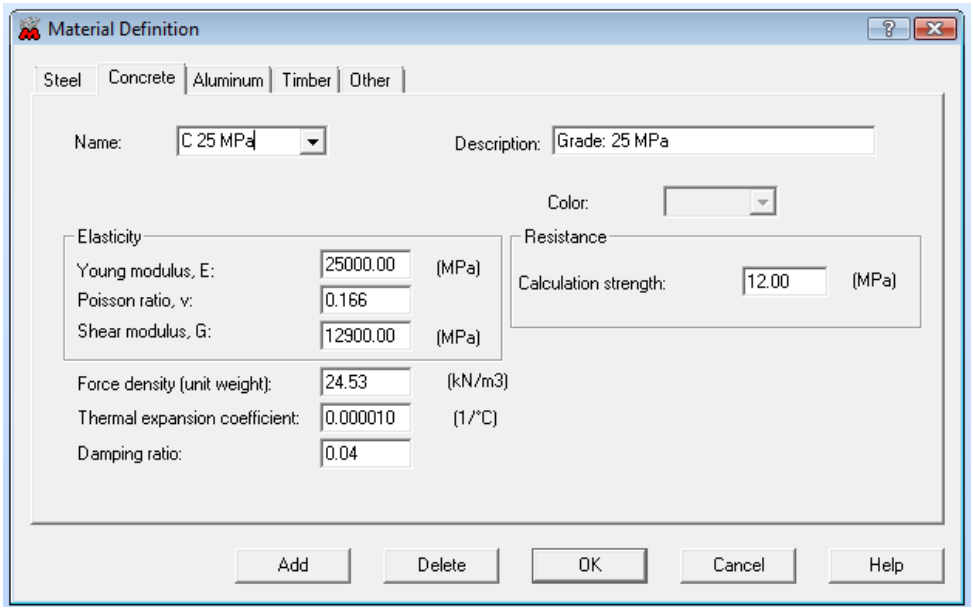
ចូល Manu ជ្រើសរើសយក Job Reference.. ឬ អ្នកអាចចុច Mouse លើពាក្យ [m][KN][Deg] ដែលស្ថិតនៅផ្នែកខាងក្រោម (ស្តាំ) ។



Materials:

- កន្លែង Materials ជ្រើសរើសយក Eurocode
- រួចចុច Modification ដើម្បីកែលក្ខណៈរបស់ដែក





www.yerngshare.blogspot.com

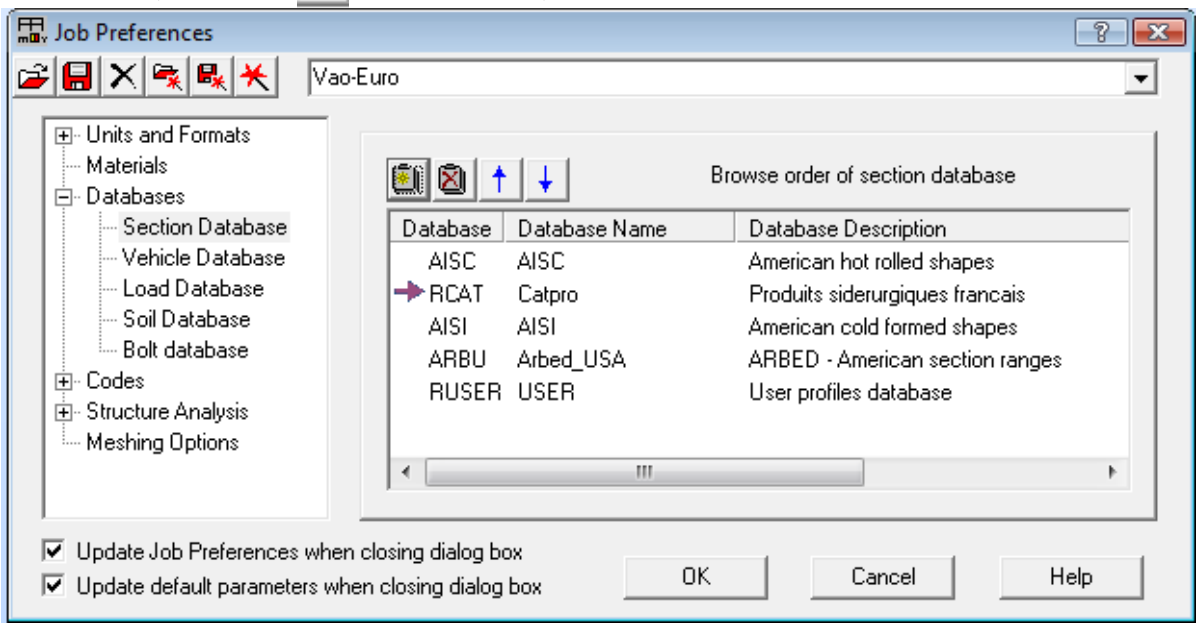
ការកំណត់ប្រភេទដែកដែលត្រូវយកមកប្រើ:

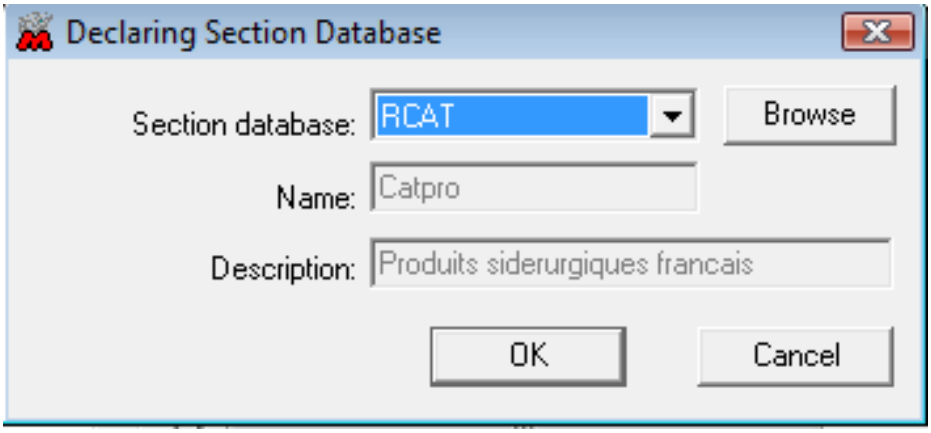
- បំពេញតំលៃដូចរូបខាងស្តាំ
- និង ចុច Add.
- កន្លែង Steel: Steel420MPa ដោយ 2D truss ជាប្រភេទដែក ដូចនេះ Concrete, Aluminum and Timber មិនទាក់ទងឡើយ ។

-ចុចលើ Databases

- បន្ទាប់មកចុច Section Databases

- និង ចុចលើ Icon  ដើម្បី Import មុខកាត់ដែក

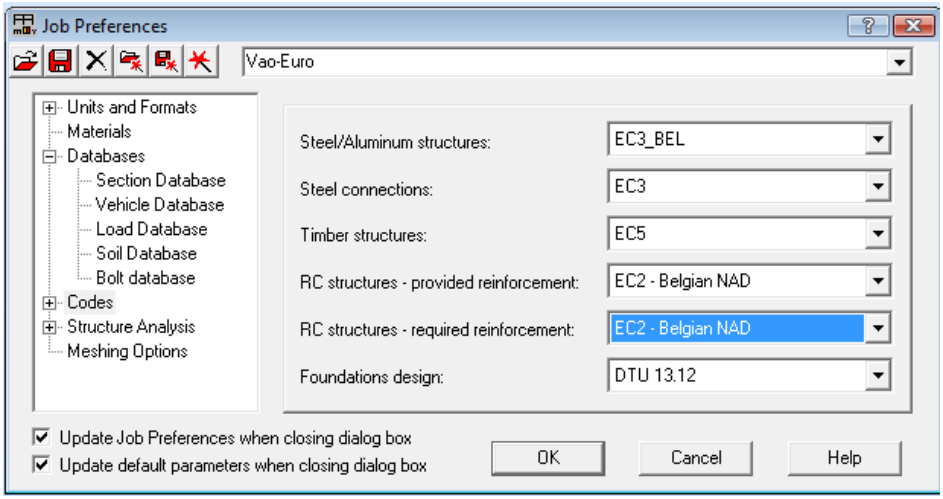




www.yerngshare.blogspot.com

ជ្រើសរើស Code:

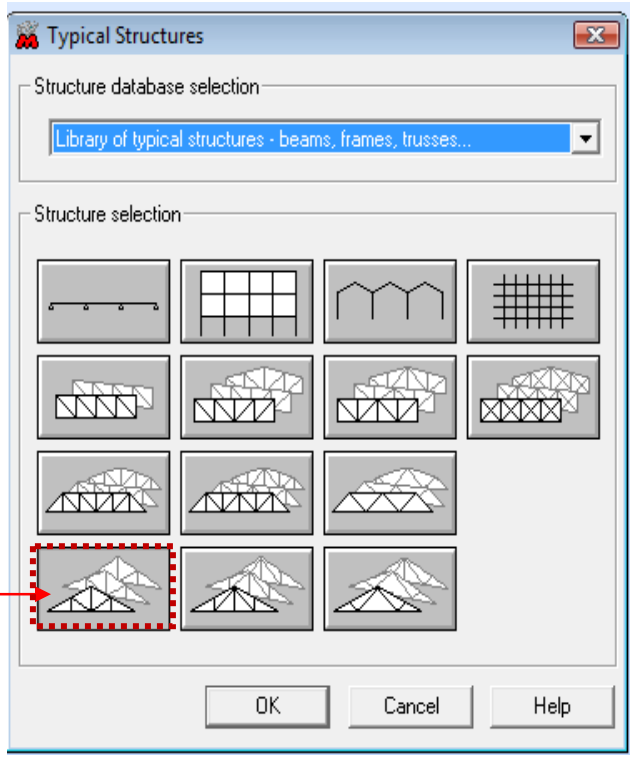
- Steel / Aluminum structure: ជ្រើសរើស EC3_BEL
- Steel connections: EC3



- Click Icon  (Save Job Preferences) បន្ទាប់មកនៅក្នុងប្រអប់ File Name អោយឈ្មោះ Eurocode រួច Click Button Save រួច OK ដើម្បីបញ្ចប់តារាង Job Preference ។

២. ការកំណត់ទំរង់របស់ Truss:

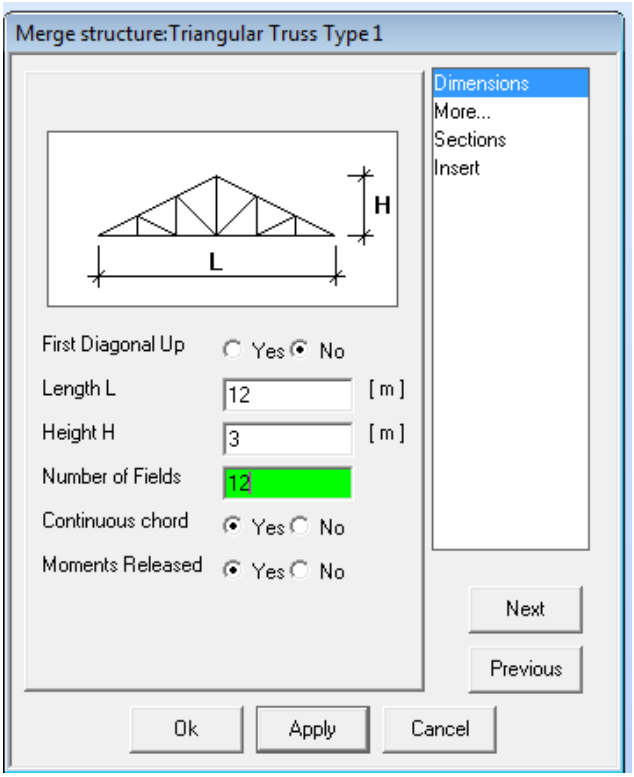
ដើម្បីកំណត់ទំរង់របស់ Truss ត្រូវចុចយក Icon  (Library structure) ដែលស្ថិតនៅពីខាងស្តាំដៃ ។



Double Click

www.yerngshare.blogspot.com

Dimensions:



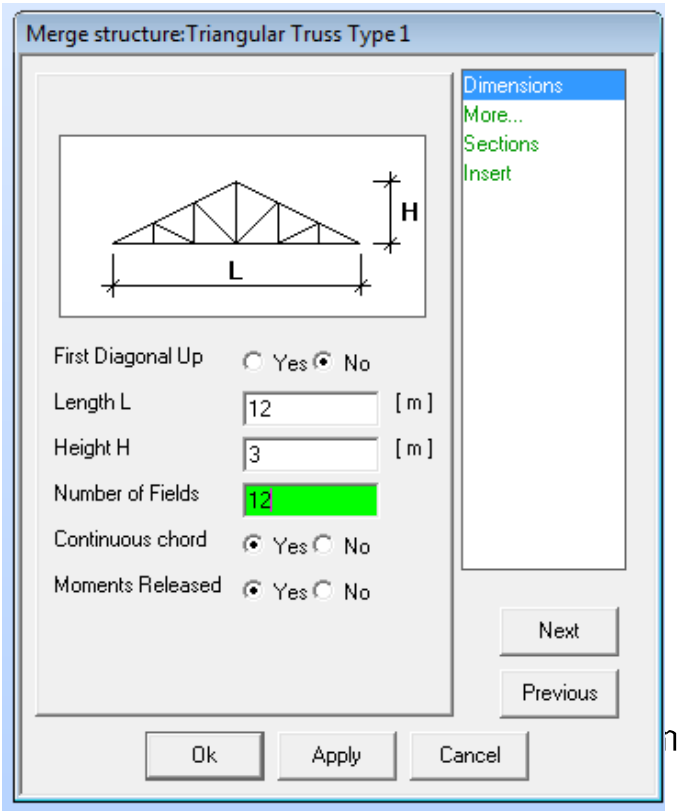
- Firse Diagonal up ចុច No
- Length L ប្រវែងរបស់ Truss=12
- កំពស់របស់ Truss = 3m
- ចំនួនប្រលោះ Number of fields=12
- Continuous chord Click Yes
- Moment Released Click yes
- បន្ទាប់មក Click Next

Sections:

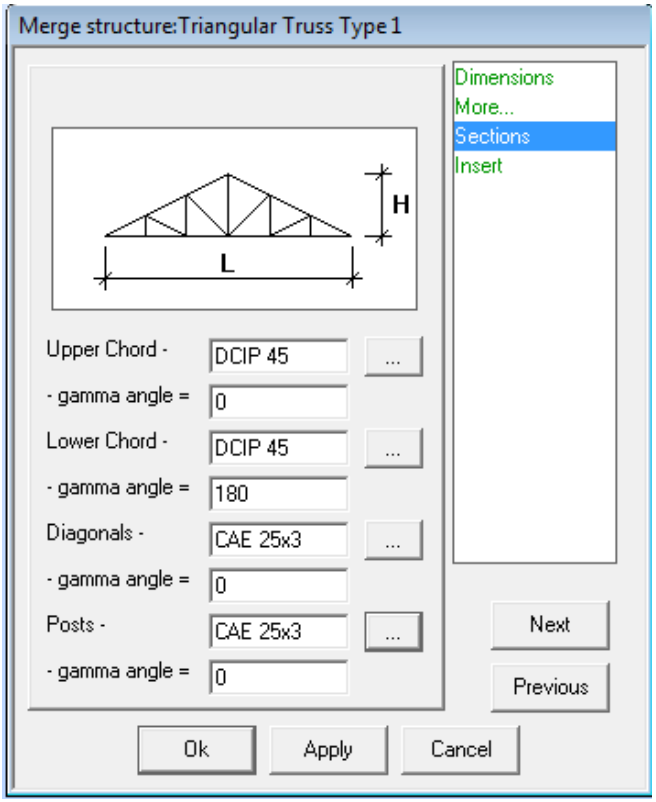
- Upper and Lower Chord = DCIP 45
- Diagonals and Posts = CAE 25 x 3
- Click Next

Insert:

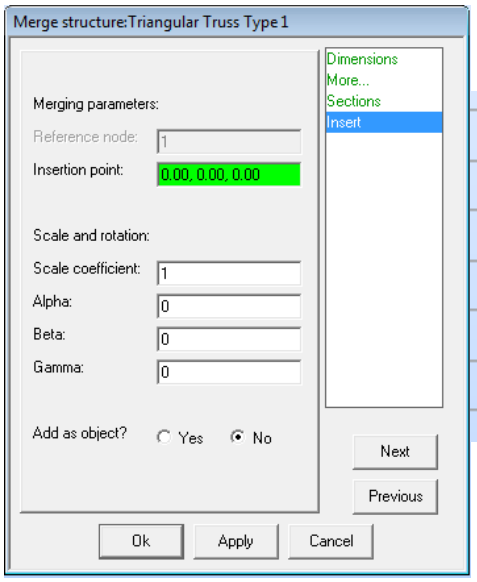
- Click នៅក្នុងប្រអប់ Insert Point: (0.00,0.00,0.00)
- Click OK

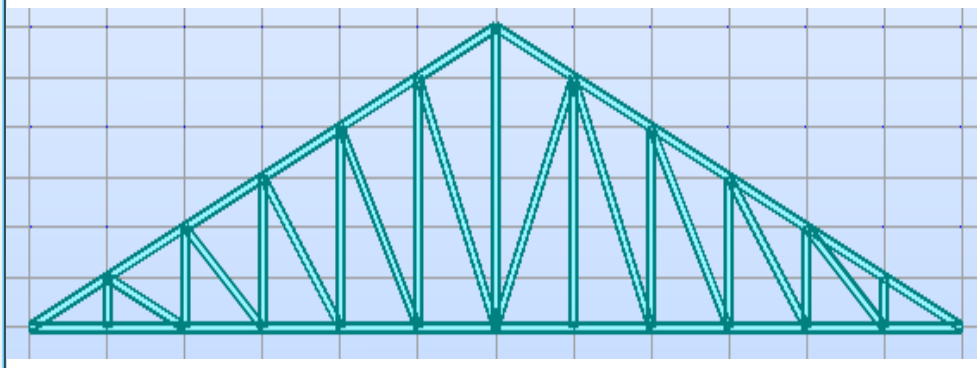


ាម:



www.yerngshare.blogspot.com

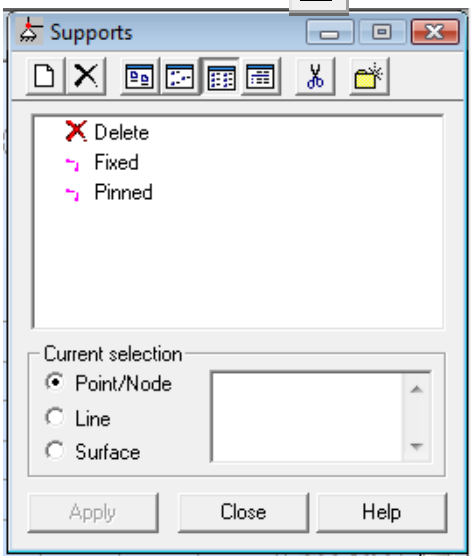


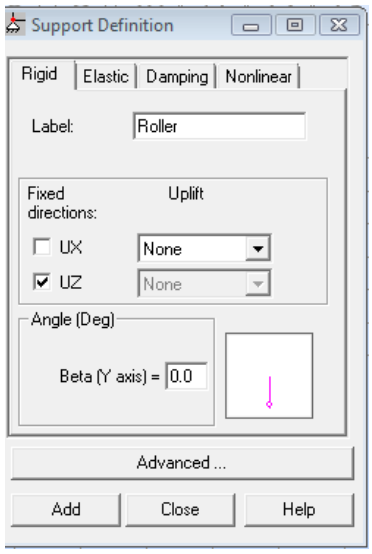


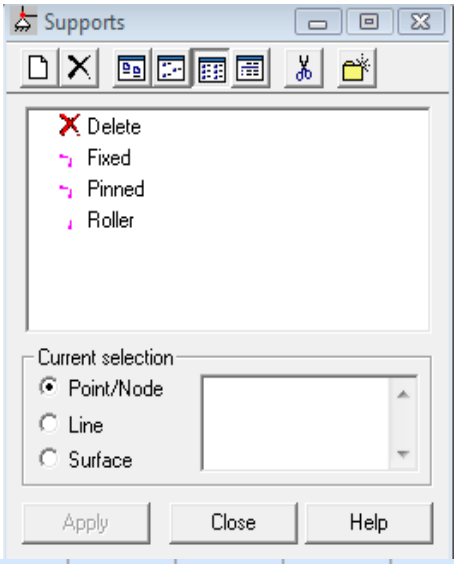
៣. *ការកំណត់ទំរ Truss:*

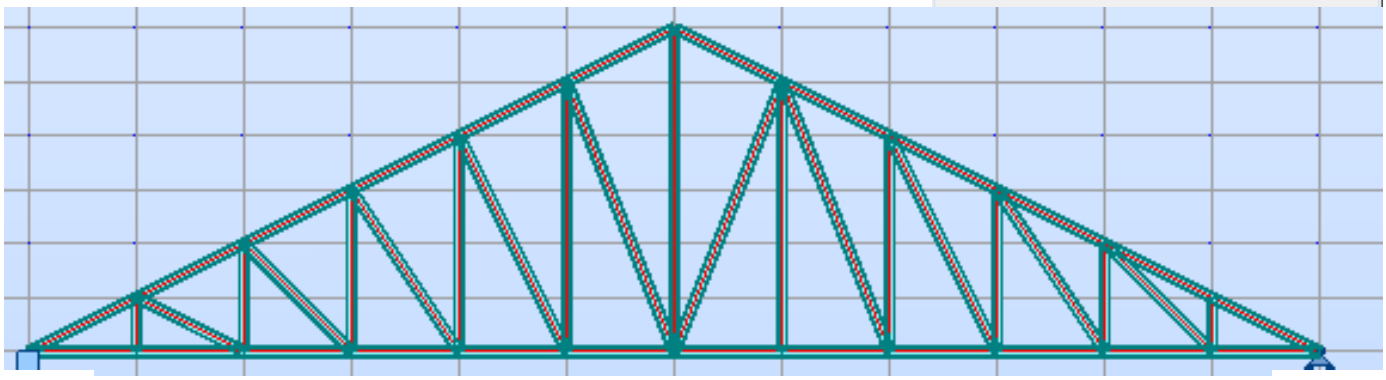
ដើម្បីកំណត់ទំររបស់ Truss អ្នកត្រូវចុចលើ Icon  (Supports)

ចុចលើ Icon  (New support definition)ដើម្បីបង្កើតប្រភេទទំរ Roller > ចុច Add.









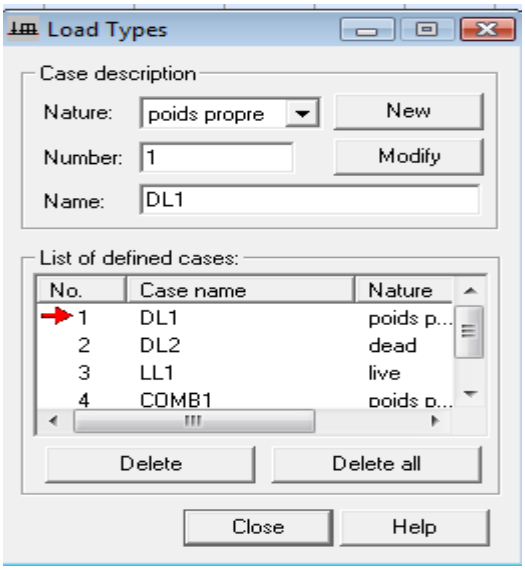
Pinned Roller

www.yerngshare.blogspot.com

<u>៤.ការកំណត់បន្ទុកដែលមានអំពើលើ Truss</u>:

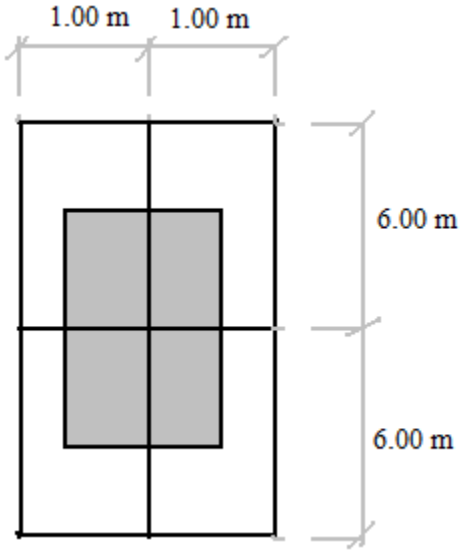
៤.១. ការកំណត់ឈ្មោះបន្ទុកដោយចុចលើ Icon  (Load types)

- Self – weigth សរសេរក្នុងប្រអប់ Name
- Click button New
- Nature : dead
- Number: 2
- Name: DL1
- Click button New
- Nature : Live
- Number: 3
- Name: LL1
- Click button New
- Close



៤.២. ការកំណត់តំលៃបន្ទុក

- Steel weight
- Insulation + Steel sheet : 0. 30 KN/m²
- Rafter : 0.30 KN/ m²
- Live Load : 0.70 KN/m²



DL = 6 m x 1 m x (0.25 + 0.30) KN/m² =3.3KN
LL = 6 m x 1 m x0.70 KN/m² = 4.2 KN

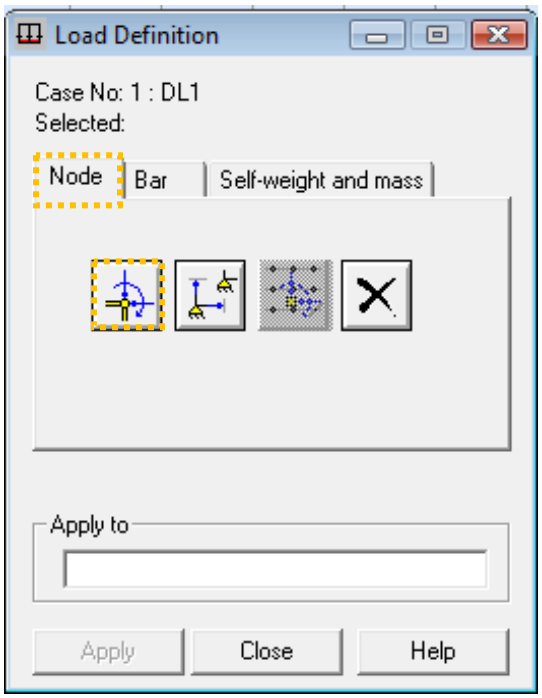
៤.៣. ការដាក់បន្ទុក DL1

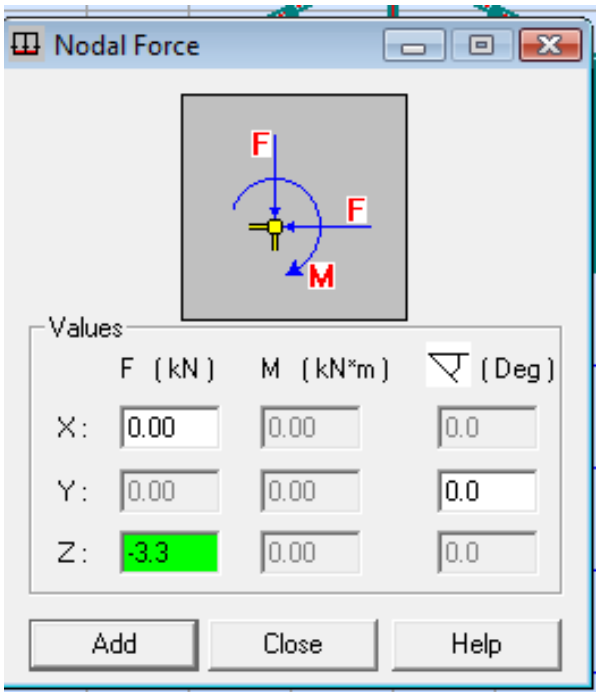
- ចុច Icon  (Load definition) ដើម្បីដាក់តំលៃបន្ទុក រួចចុចប្តូរប្រភេទបន្ទុកទៅជា DL1

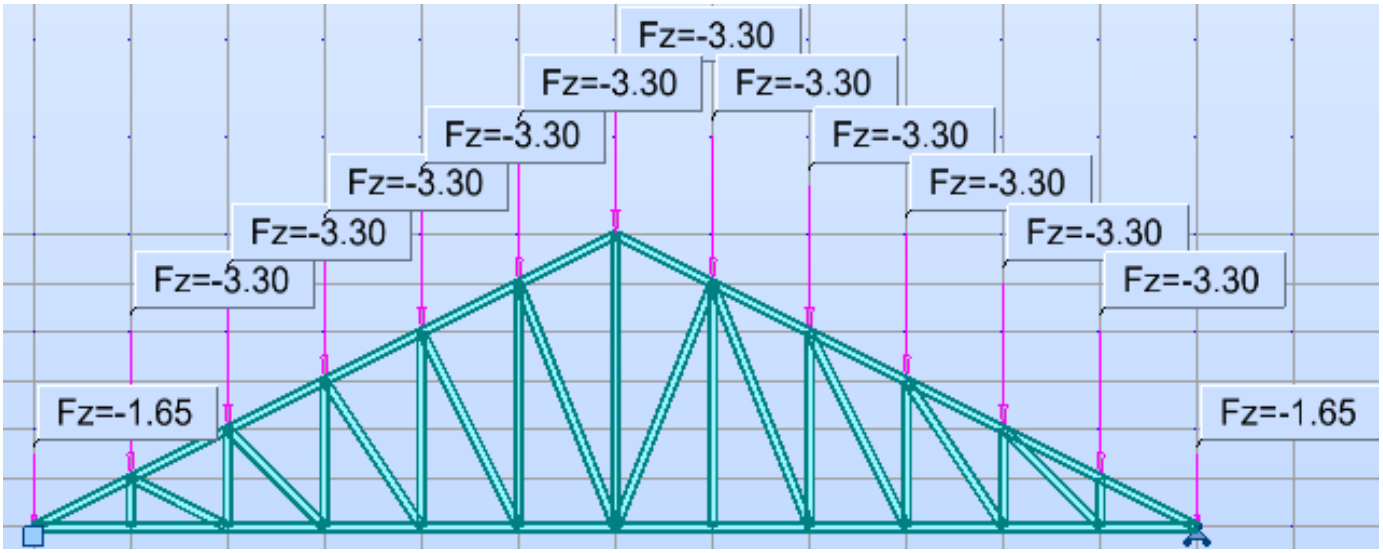


- ចុច Node
- បំពេញតំលៃ Z = - 3.3
- ចុច Add

www.yerngshare.blogspot.com

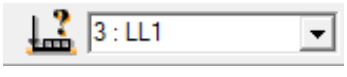






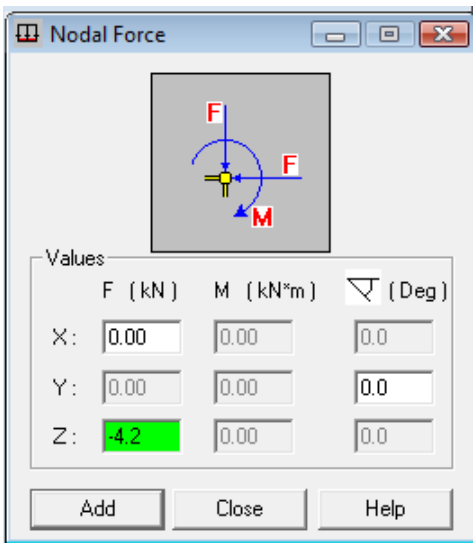
៤.៤. ការដាក់បន្ទុកអថេរ LL1

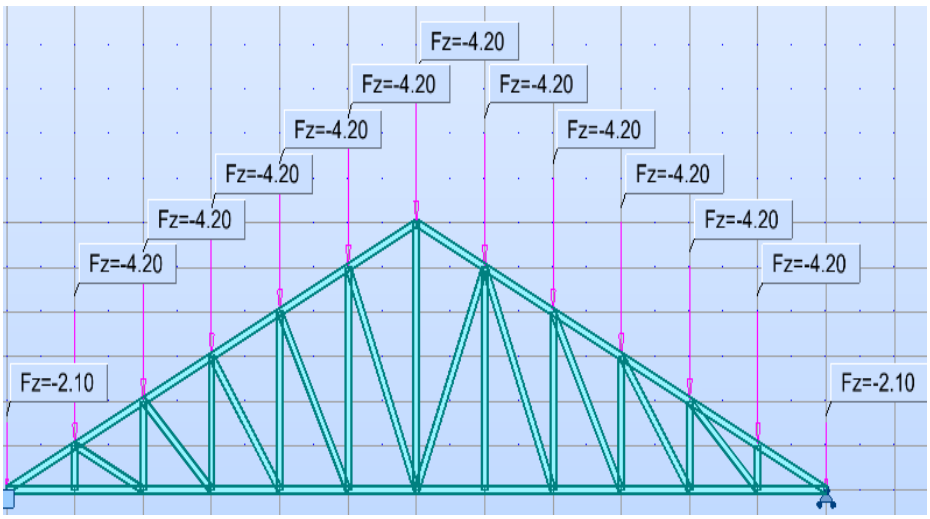
របៀបធ្វើដូចខាងលើដែរ:



- ប្តូរប្រភេទបន្ទុកជា LL1
- បំពេញតំលៃ Z = - 4.2
- ចុច Add

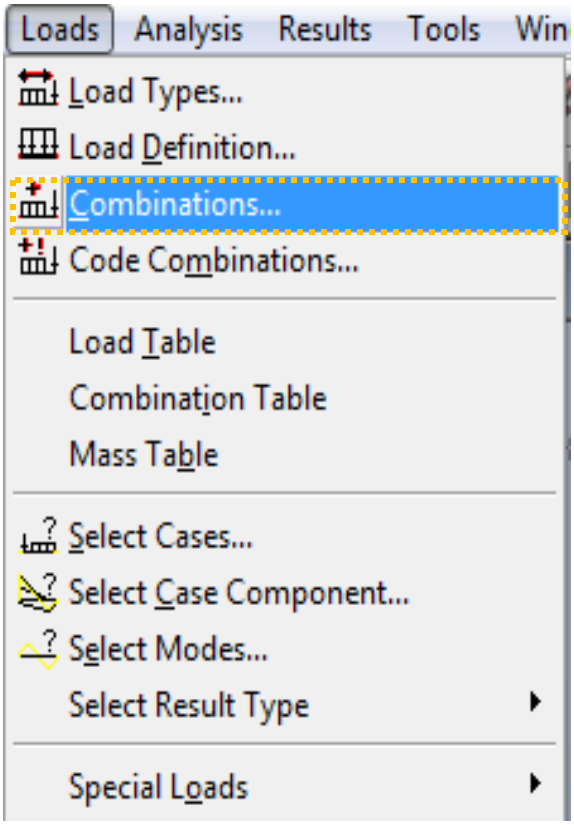
www.yerngshare.blogspot.com

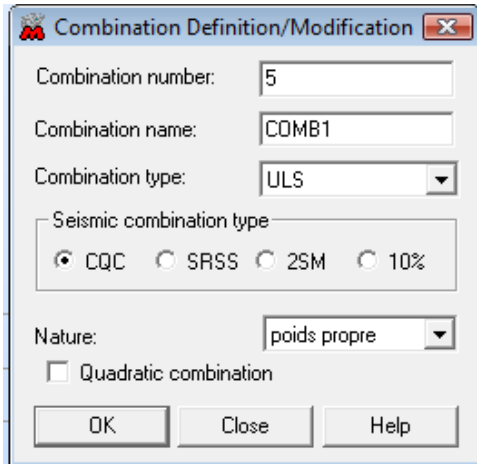


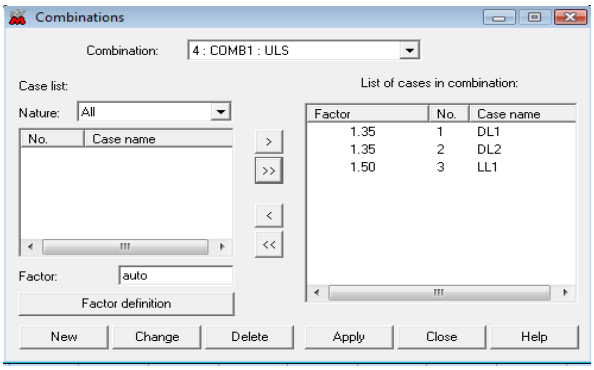


៤.៥. ការបន្សំបន្ទុក (Load combination)

- ចុចលើ Loads menu > Combinations
- Combination name: COMB1
- Combination Types: ULS
- Clic k button OK.
- Click Icon >> (Next)



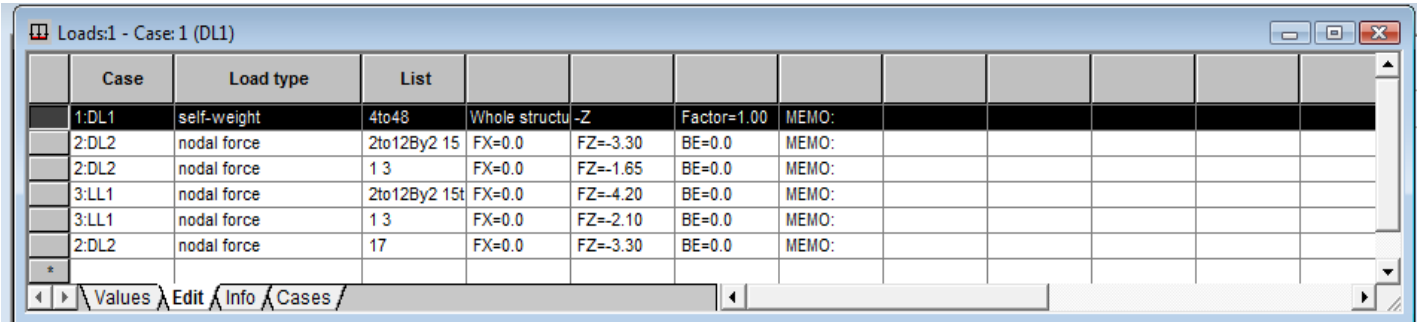




www.yerngshare.blogspot.com

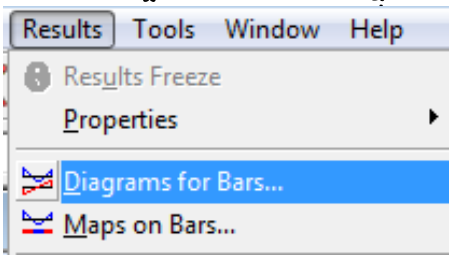
ចំណាំៈ ពេលដាក់បន្ទុករួចហើយ យើងត្រូវ Check មើលក្នុងតារាងបន្ទុក

របៀបចូលៈ ចុច Loads menu > Load Table

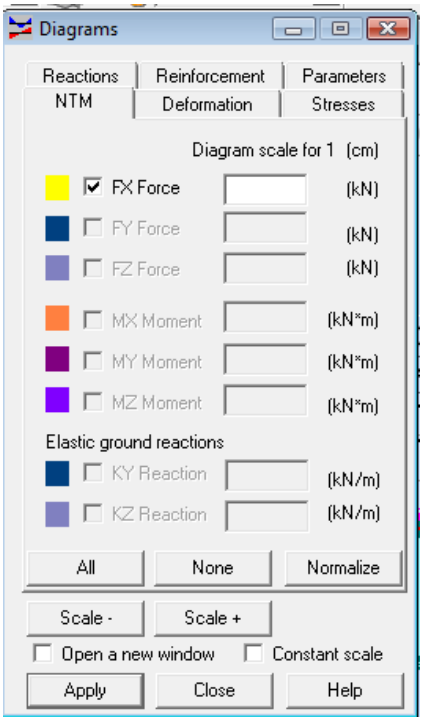


៥. ការគណត់រកកំលាំងក្នុងរបាររបស់ Truss:

ចុច Icon (Calculations) ដើម្បីគណនារបស់កំលាំងក្នុងរបស់របារនិមួយៗ នៅពេលមើលលទ្ធផលត្រូវចុច Results > Diagrams for bars…..



- Tick FX Force ដើម្បីមើលកំលាំងចំអ័ក្ស (Axial Force) របស់របារនិមួយៗ
- ចុច Button Apply

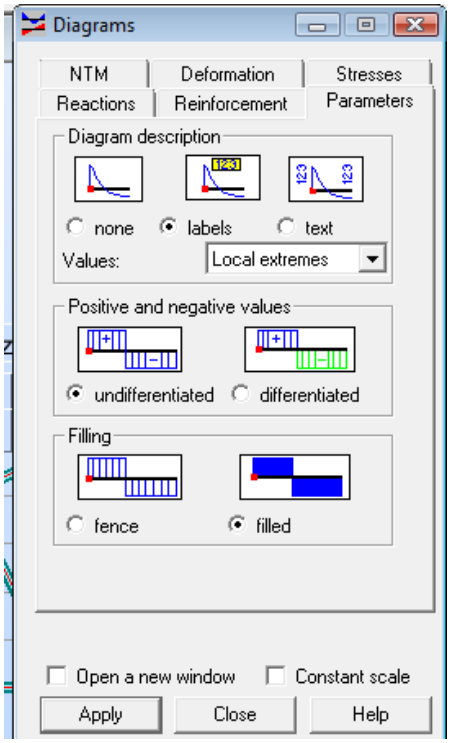


ដើម្បីមើលតំលៃ

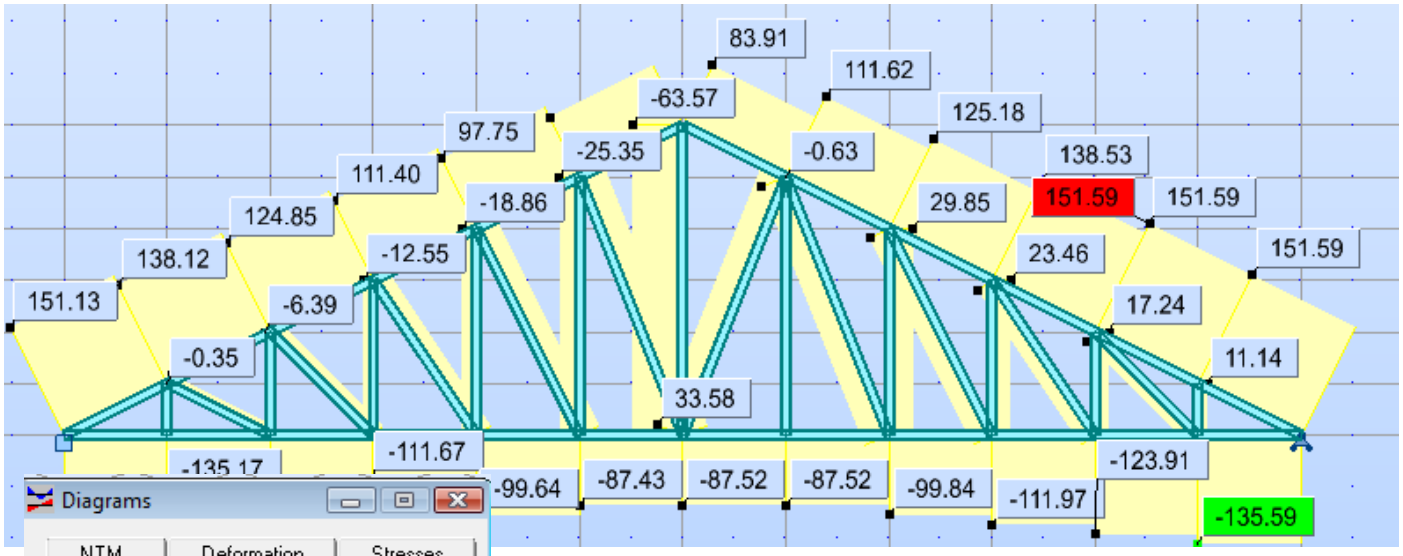
នៅលើ Diagrams

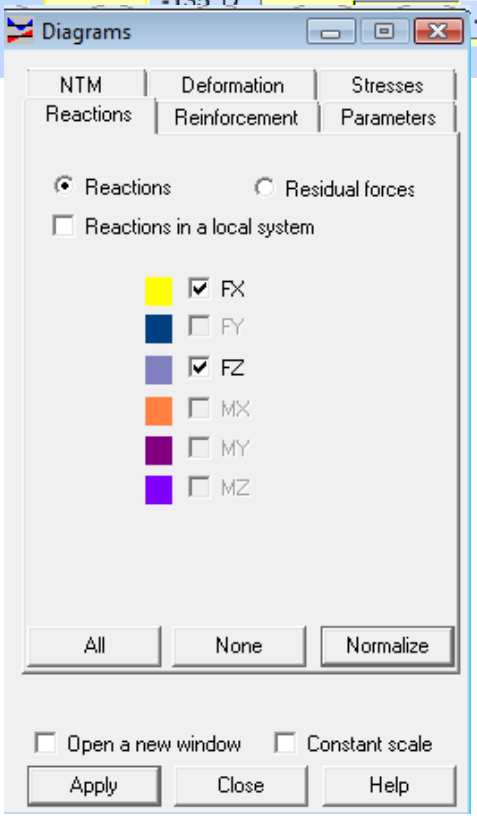
ត្រូវចុចលើ Parameters

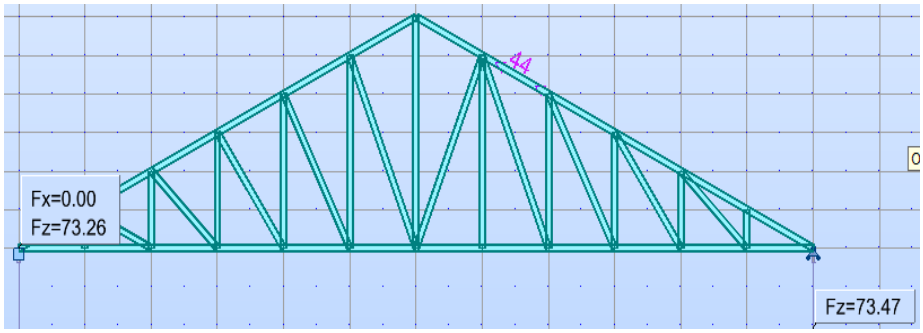
រួចចុចយក Labels > Apply



www.yerngshare.blogspot.com



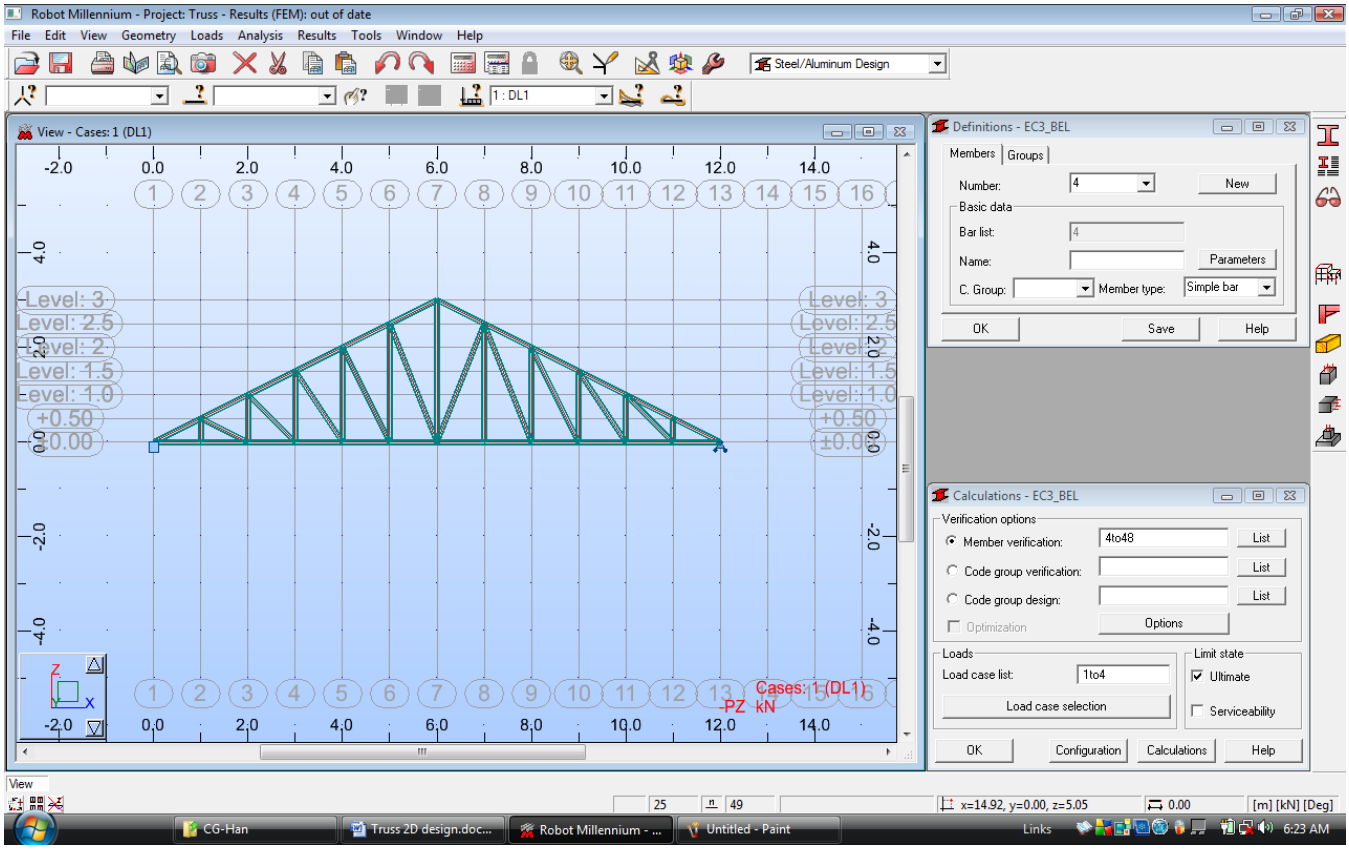




www.yerngshare.blogspot.com

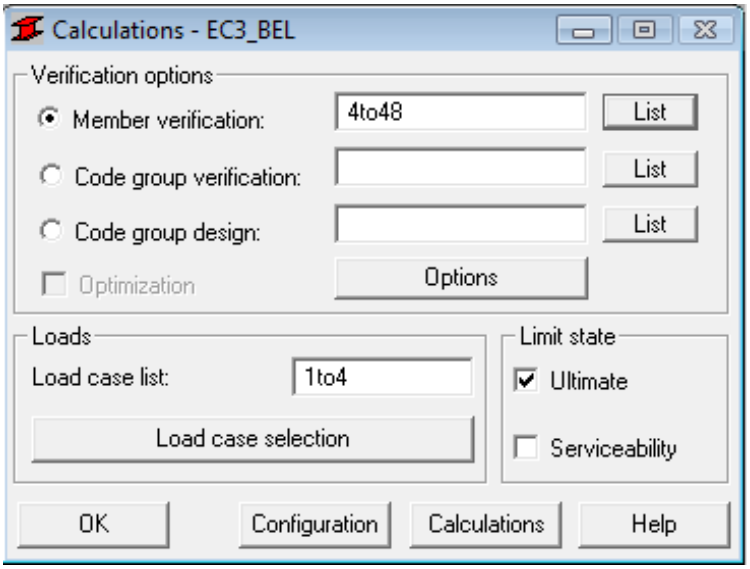
ខ.ការត្រួតពិនិត្យ (Check) ឬ Design អ្នកគាត់រោងរបស់ Truss:

ចូរទៅក្នុង Analysis > Bar structure design > Steel / Aluminum members design > Definition / Calculations set…

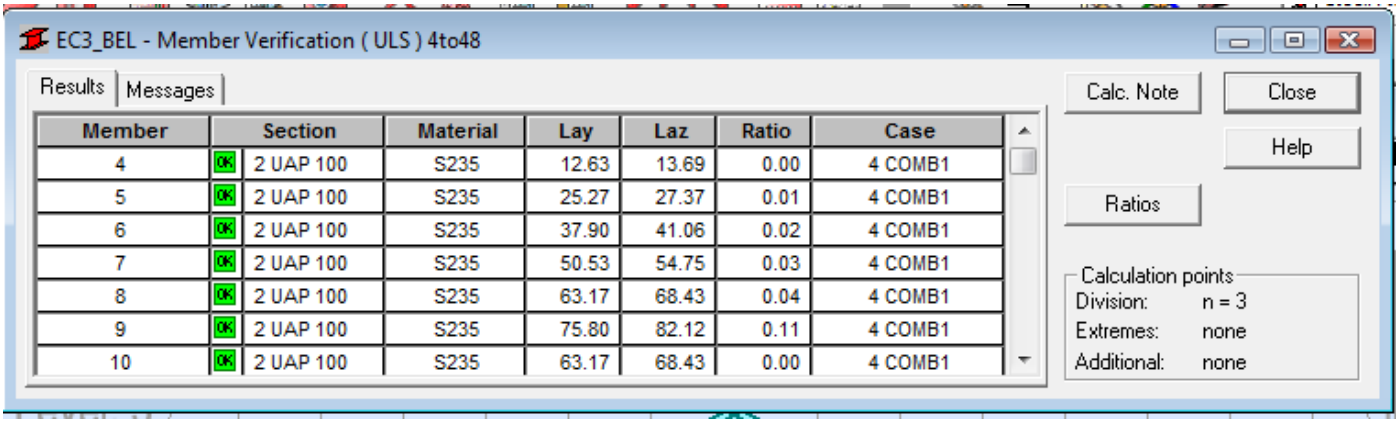


ខ.១ Member verification:

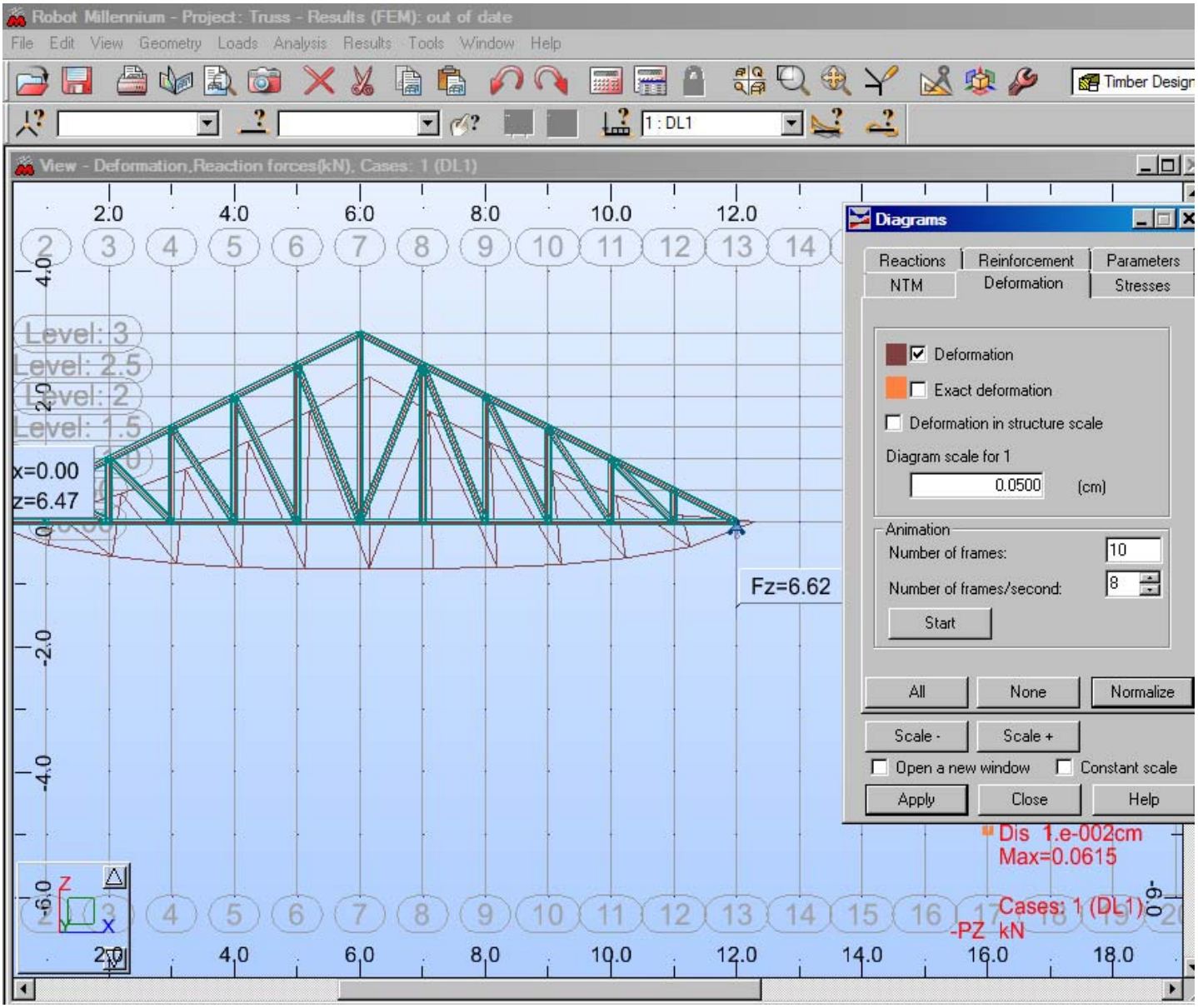
ត្រង់ Verification options click លើប្រអប់ member verification: 4 to 48 > Calculations



www.yerngshare.blogspot.com

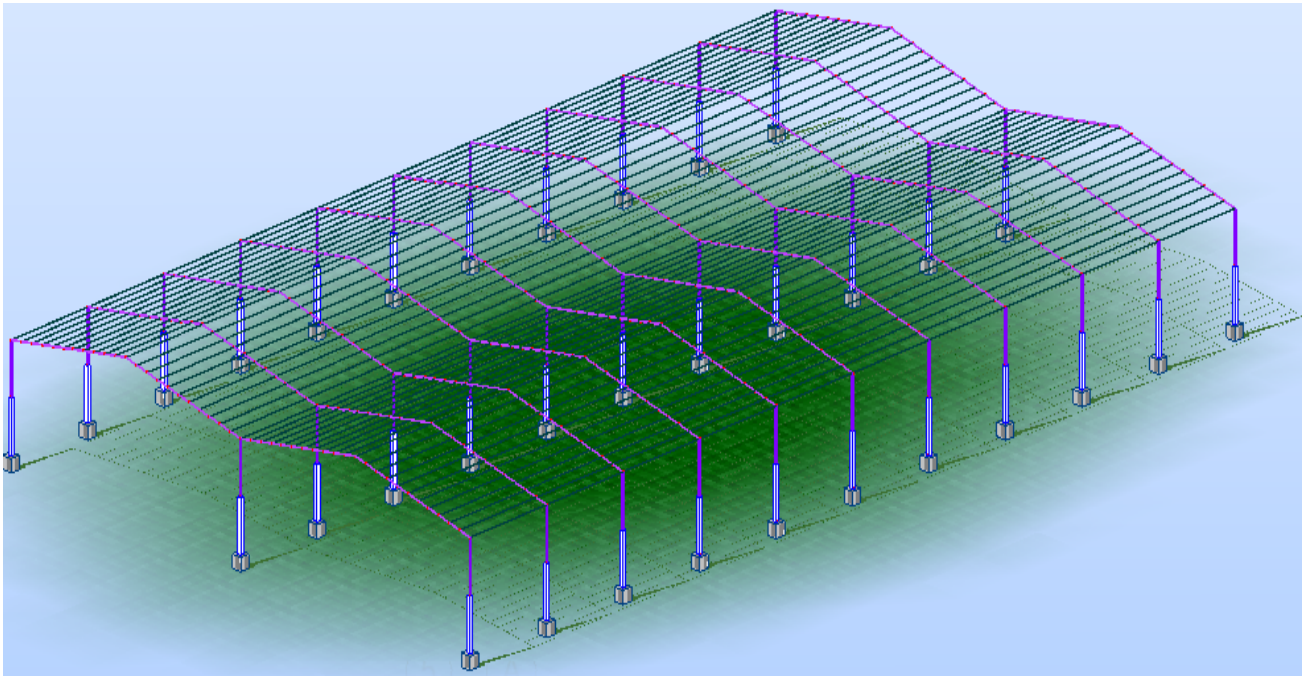


Member	Section	Material	Lay	Laz	Ratio	Case
4	OK 2 UAP 100	S235	12.63	13.69	0.00	4 COMB1
5	OK 2 UAP 100	S235	25.27	27.37	0.01	4 COMB1
6	OK 2 UAP 100	S235	37.90	41.06	0.02	4 COMB1
7	OK 2 UAP 100	S235	50.53	54.75	0.03	4 COMB1
8	OK 2 UAP 100	S235	63.17	68.43	0.04	4 COMB1
9	OK 2 UAP 100	S235	75.80	82.12	0.11	4 COMB1
10	OK 2 UAP 100	S235	63.17	68.43	0.00	4 COMB1

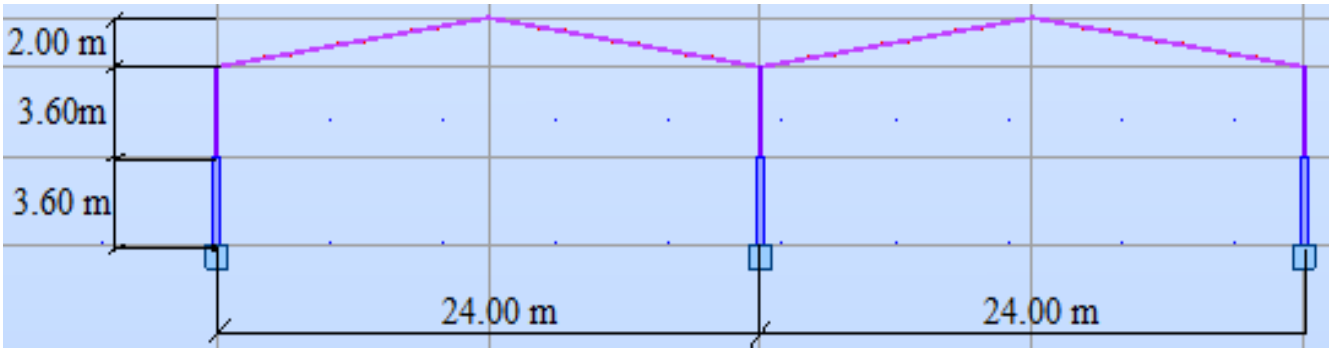


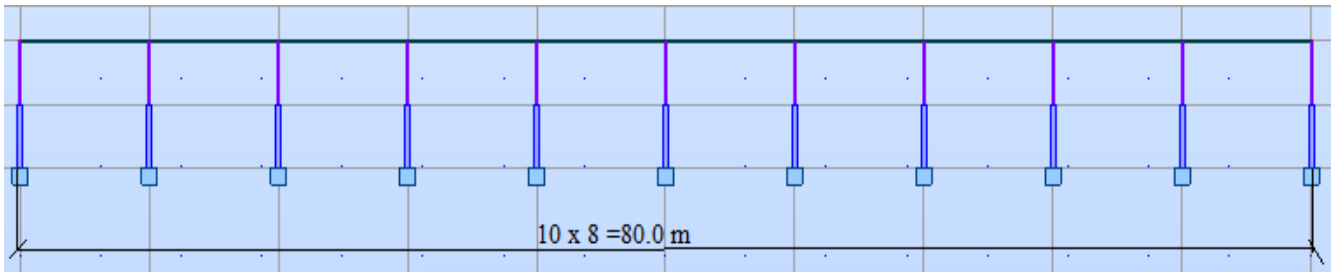
www.yerngshare.blogspot.com

មេរៀនទី៧:3D Steel Frame



3D View





Elevation View

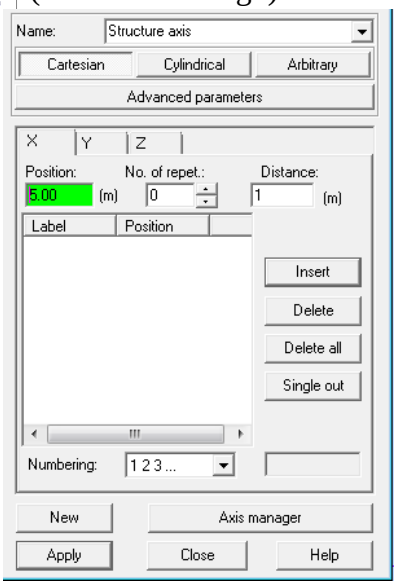
1. Section :
   - R C Column (30 x 30) cm
   - UPE: 160
   - CAE: 60 x 5
   - CAE: 40 x 3
2. Material of Properties:
   - Compressive strength of concrete(Cube) = 25 MPa
   - Yield strength of Longitudinal reinforcement = 400MPa
   - Yield strength of UPE , CAE Steel = 355 MPa
2. Loads:
   - Selfweigth
   - Insulation + Steel sheet :0.30 KN/m²
   - Rafter : 0.5 KN/m²
   - Live Load : 0.35KN/m²

១៩ ទីជម្រកទន្លេ

១. Click Program Robot Millennium បន្ទាប់មក Click Icon (Frame 3D Design)

២.ការកំណត់អ័ក្ស Axis សំរាប់គួរ

ចូរ Click Icon (Axis Definition) ដើម្បីបង្កើត Axis

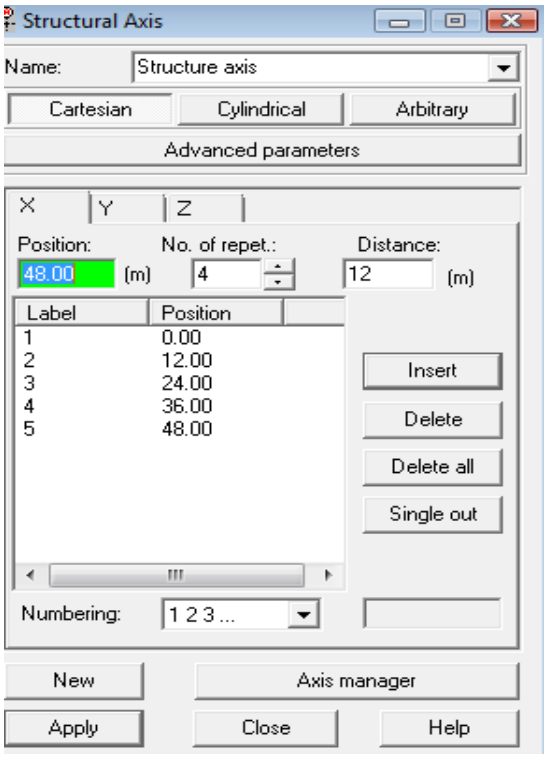


២.១ កំណត់អ័ក្ស X:

ត្រង់អ័ក្ស X : បំពេញតំលៃដូចខាងក្រោម:

- Position : 0.00
- No. of repet : 4.00
- Distance : 12.00
- Numbering :123..

Click Command button Insert

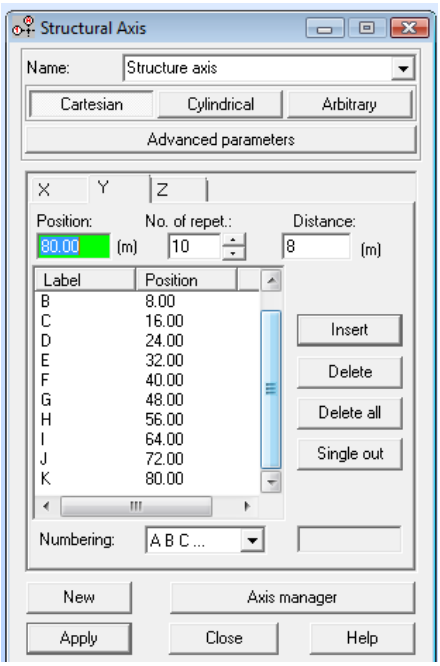


២.២. កំណត់អ័ក្ស Y:

Click លើអ័ក្ស Y រួចកន្លែង Numbering ប្តូរពី 123.. ទៅជា ABC….

- Position : 0.0
- No. of repet : 10
- Distance : 8.0

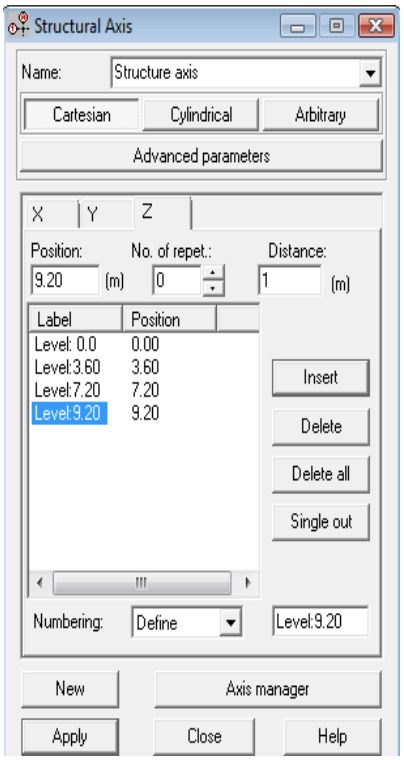
Click Insert



www.yerngshare.blogspot.com

២.៣ កំណត់អ័ក្ស Z:

Click លើអ័ក្ស Z រួច Click កន្លែង Numbering: Define



- Position : 0.0
- Leve : 0.0
- No. of repet :0.0
- Distance : 1

} Click Command button Insert

- Position :3.60
- Level : 3.6
- No. of repet : 0.0
- Distance : 1

} Click Command button Insert

- Position : 7.20
- Level: :7.20
- No. of level :0.0
- Distance :1

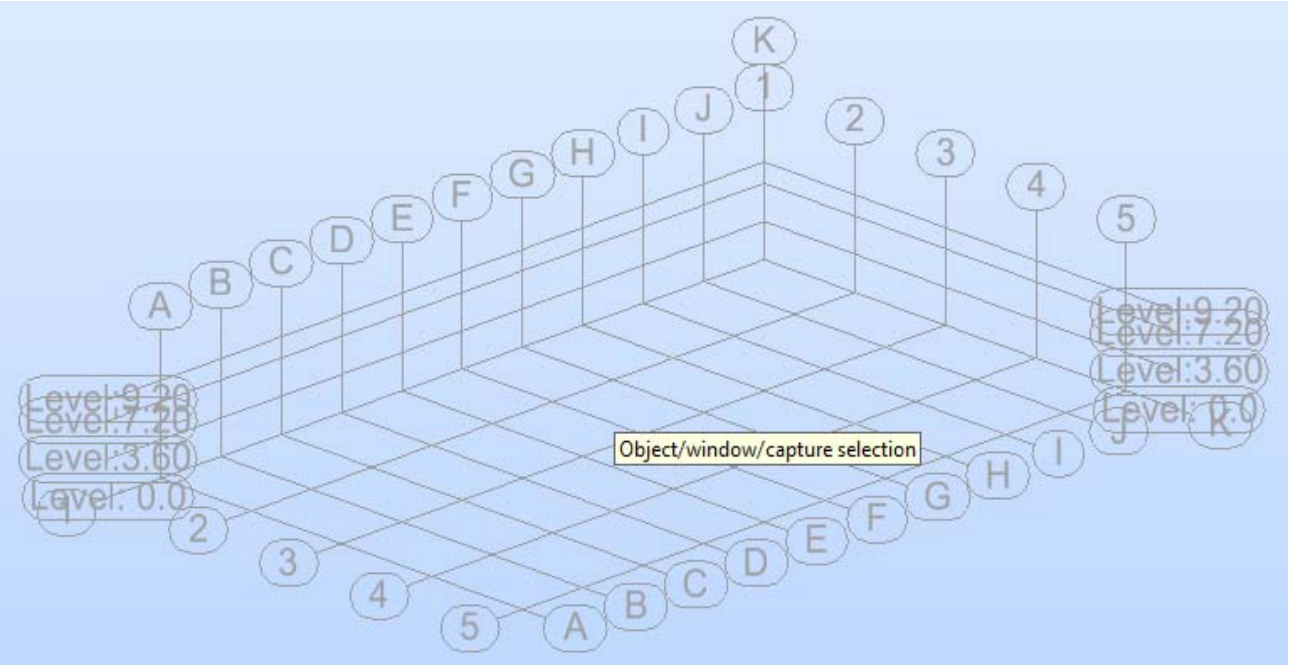
} Click Command button Insert

- Position : 9.20
- Level : 9.20
- No. of repet: 0.0
- Distance : 1

} Click Command button Insert

បន្ទាប់មក Click Command button Apply.

www.yerngshare.blogspot.com



៣. ការតួរ សសរ និង Steel Frame :

ចូរលើ Icon Bars

៣.១ ការបង្កើតមុខកាត់សសរ 30 x 30 , UPE : 160, CAE : 60 x 5, CAE : 40 x 3

- Bar types : RC Column
- Section : ចុចលើរូប ... ដើម្បីបង្កើតមុខកាត់សសរ (b =30 , h = 30)
- Section type: RC column

} Click button Add

ការបង្កើតសសរដែក

- Section type: Steel
- Database : Catpro
- Section : UPA 160

} Click Command button Add

ការបង្កើតធ្នឹមដែក

- Bar type : Simple bars
- Section : Steel
- Database : Catpro
- CAE : 60 x 5

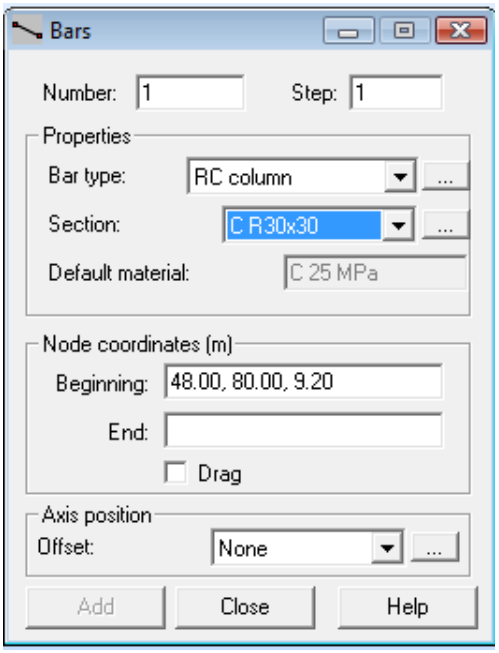
} Click Command button Add

ការបង្កើតដែរណែង

www.yerngshare.blogspot.com

- Bar type : Simple bars
- Section : Steel
- Database : Catpro
- CAE : 40 x 3

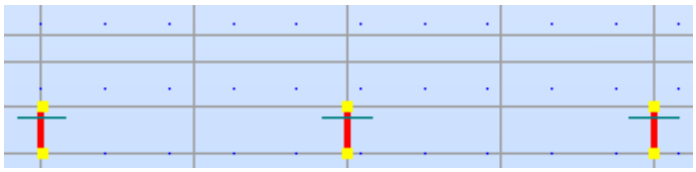
Click Command button Add



បន្ទាប់មក Click Command button Close.

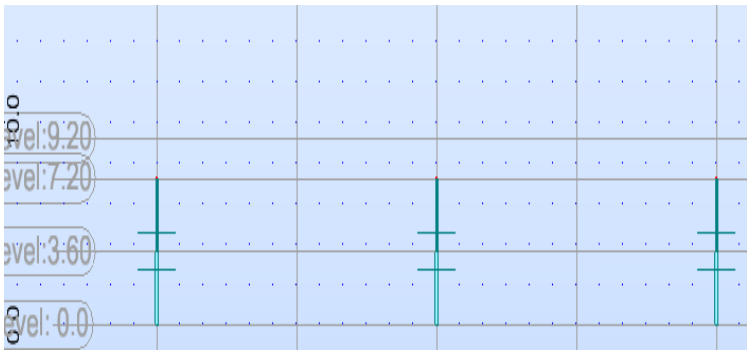
<u>៤. ការគូរសសរបេតុងអាម៉ែ(30 x 30)</u>

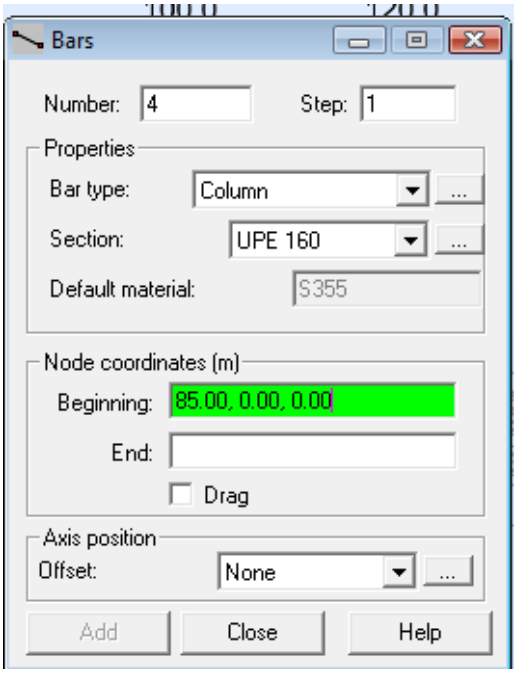
- Bar type : RC column
- Section : RC 30 x 30
- យក Mouse Click ក្នុងប្រអប់ Beginning
- រួចគូសដូបង្ហាញក្នុងរូប ខាងក្រោម



<u>៥. ការគូសសសរ UPA:160</u>

- Bar type : Column
- Section : UPA 160
- យក Mouse Click ក្នុងប្រអប់ Beginning
- រួចគូសដូបង្ហាញក្នុងរូប ខាងក្រោម



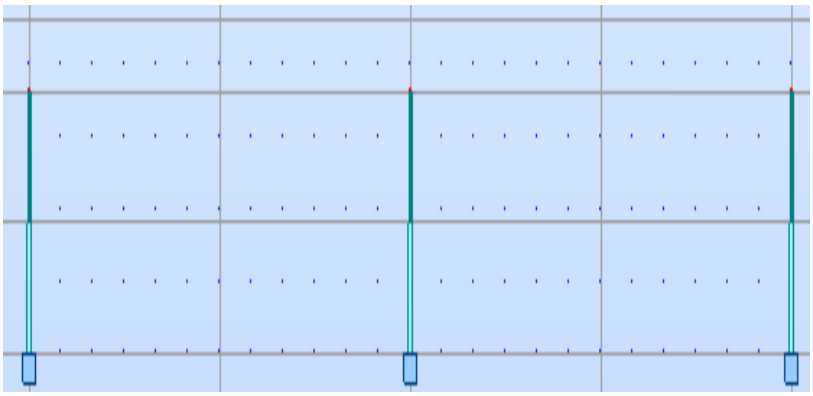


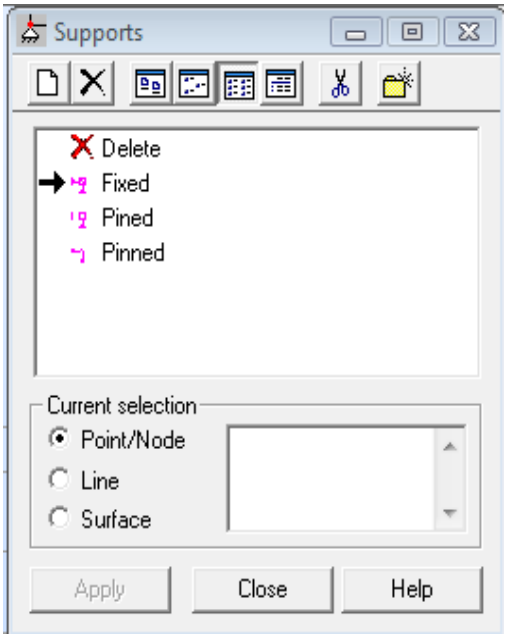
www.yerngshare.blogspot.com

<u>ខ. ការដាក់ទំរ(Support)</u>

- Click លើ Icon Support
- Type of support : Fixed
- Current selection: Point/Node

Tick លើ រូបភាព និង បង្ហាញដូចខាងក្រោម

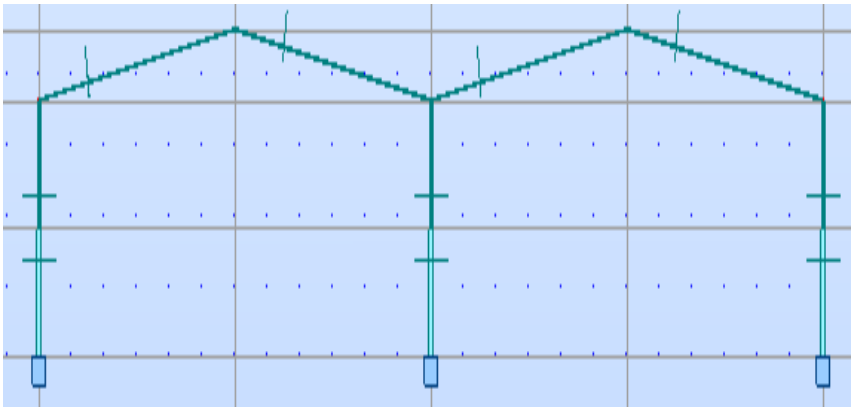


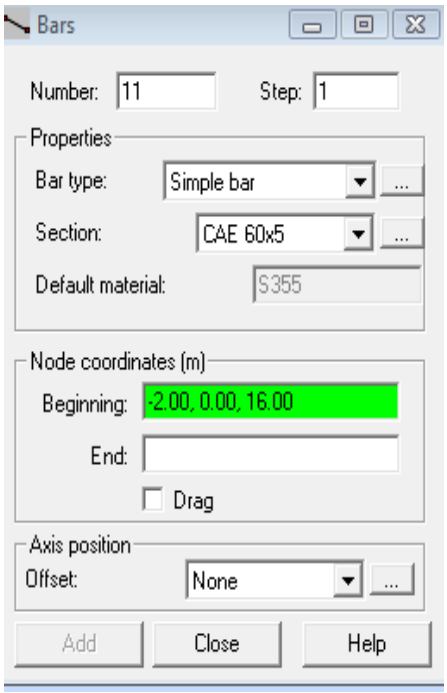


<u>ឃ. ការគូសធ្នឹម(CAE 60x5)</u>

- Bar type : Simple bars
- Section : UPA 160
- យក Mouse Click ក្នុងប្រអប់ Beginning

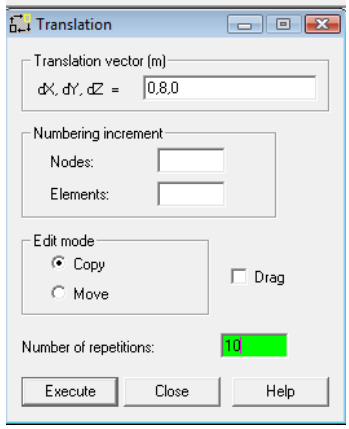
រួចគូសដូបង្ហាញក្នុងរូប ខាងក្រោម

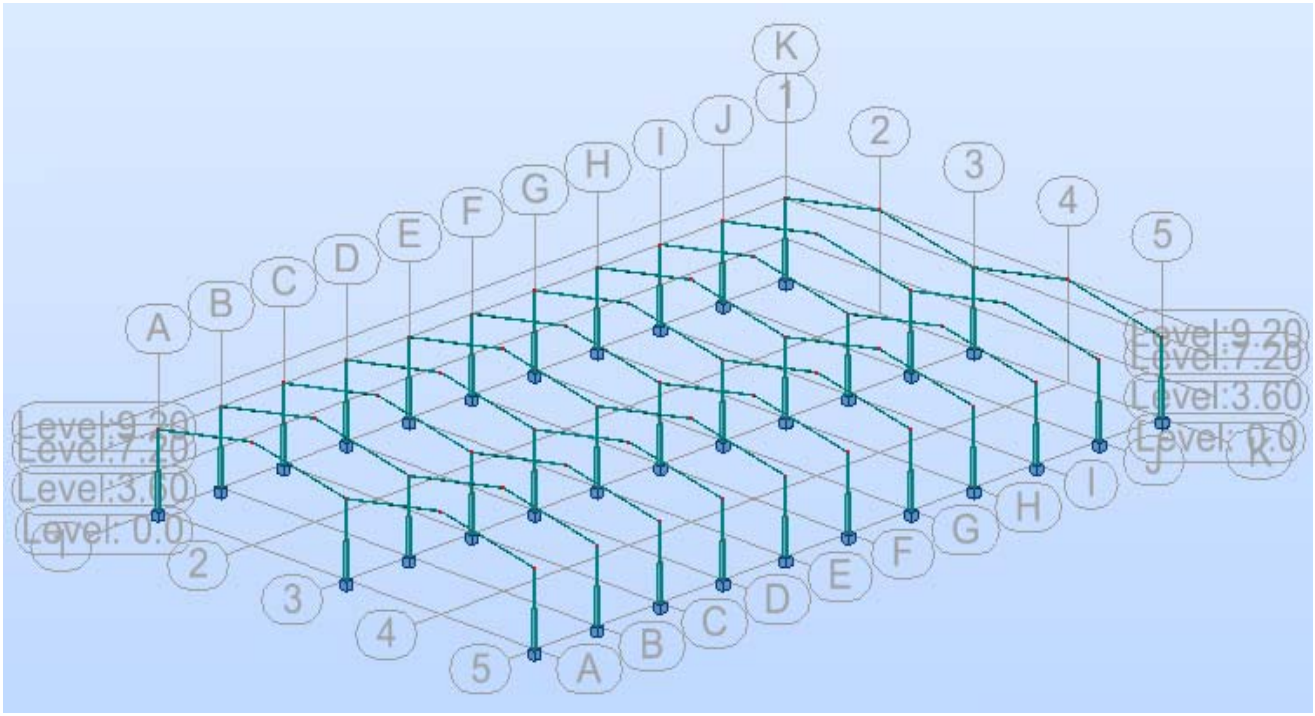




៨. ការចំលងរូបភាព (Copy Frame)

- Select All ដោយចុច Ctrl + A
- ចូរ Edit menu > Edit > Translate……
- dx, dy, dz =0,8,0
- Numbering = 10
- Click Command button Excute

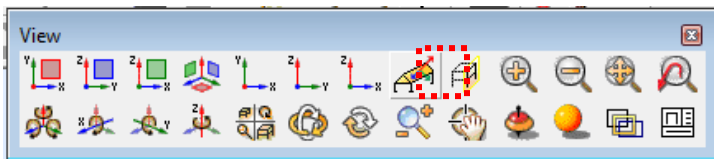


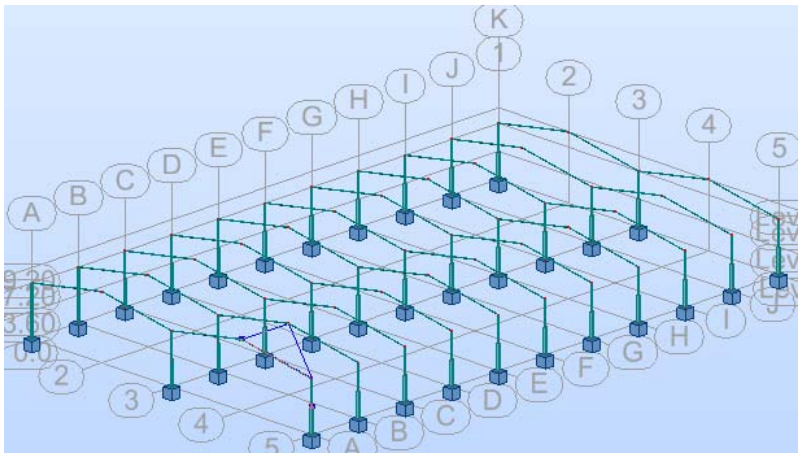


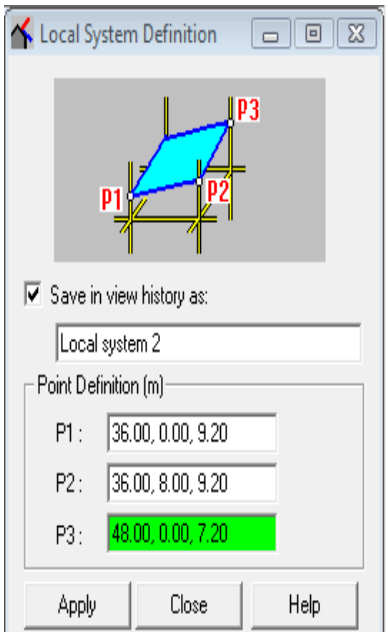
www.yerngshare.blogspot.com

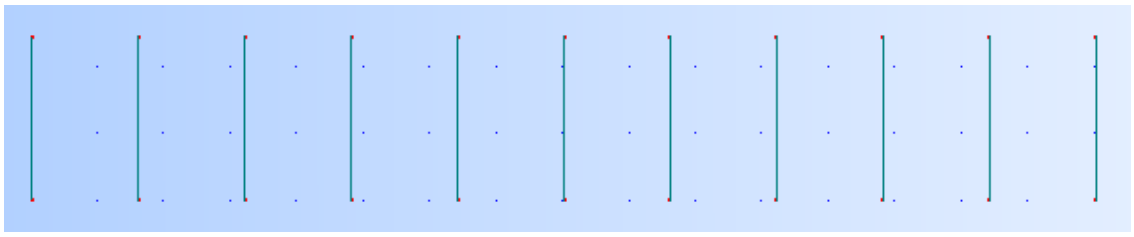
៩. ការគូស (CAE 40 x 3)

- Click Icon Local System Definition





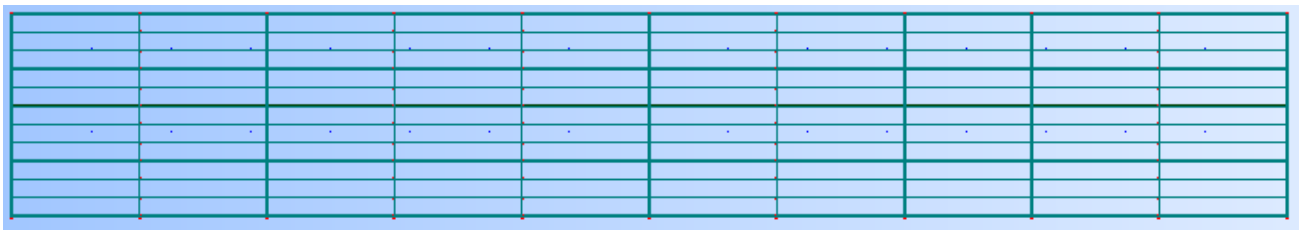




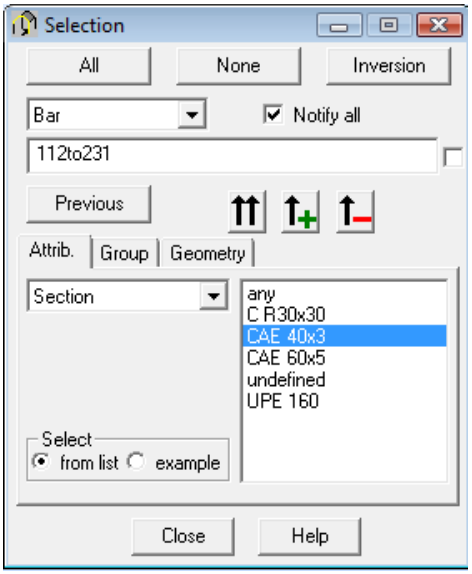
១០. គូសរបារដែលមានមុខកាត់ CAE 40 x 3

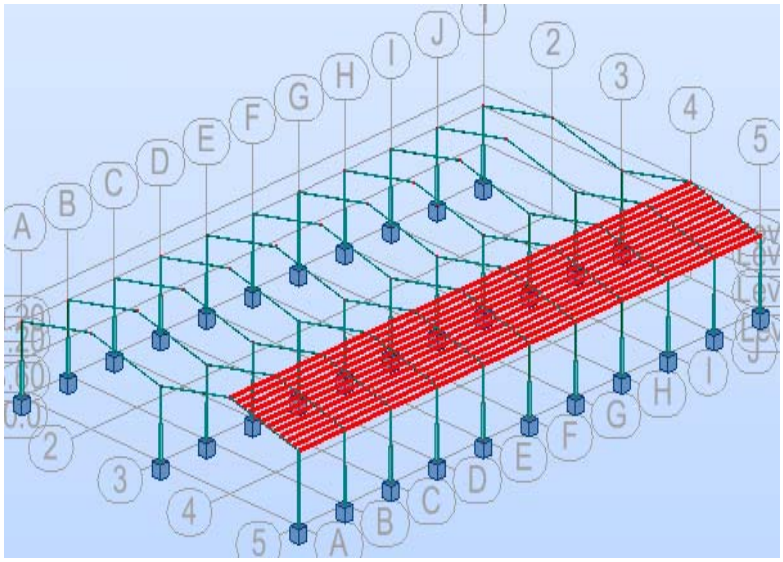
-បន្ទាប់មក Select លើដែលទើបនឹងគូស រួច Copy ដោយ:

- វាយលេខនៅក្នុង Coordinate (0,-1.1,0)
- Number of repetition :11
- Click Command button Excute

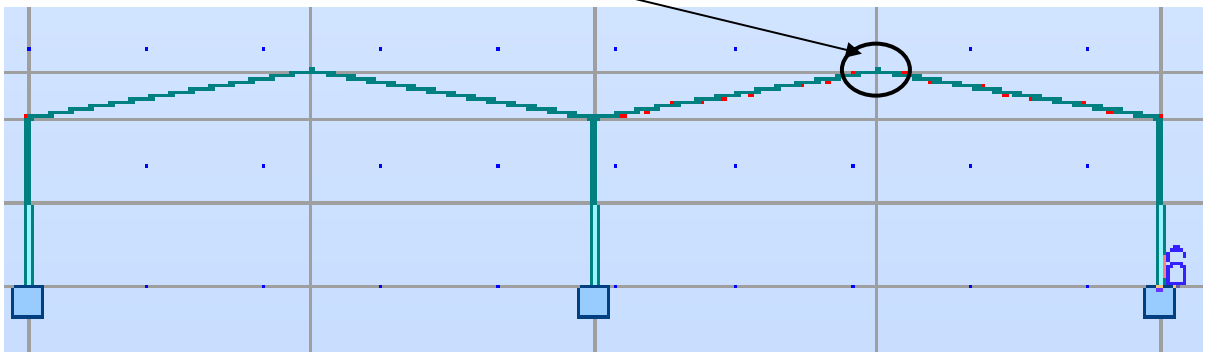


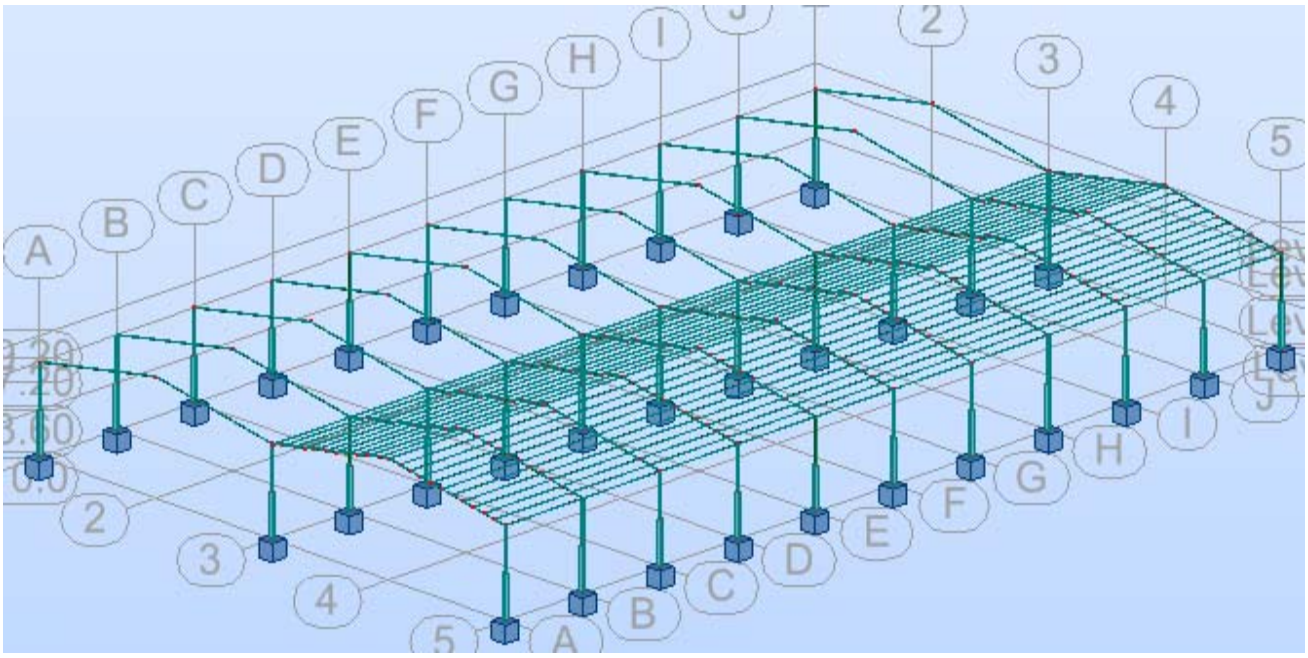
ចុច Edit menu > Edit > Vertical Mirror:





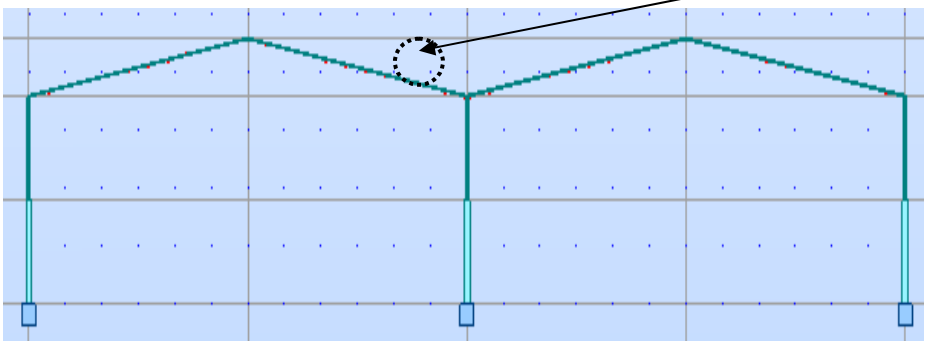
បន្ទាប់ពី Mirror រួចចុចត្រង់ចំនុច

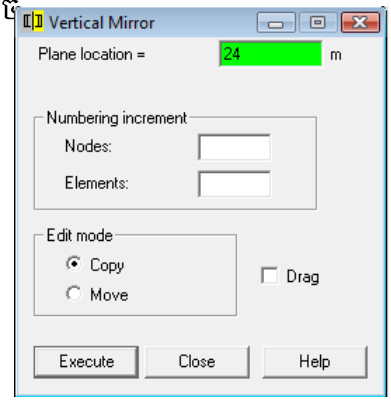


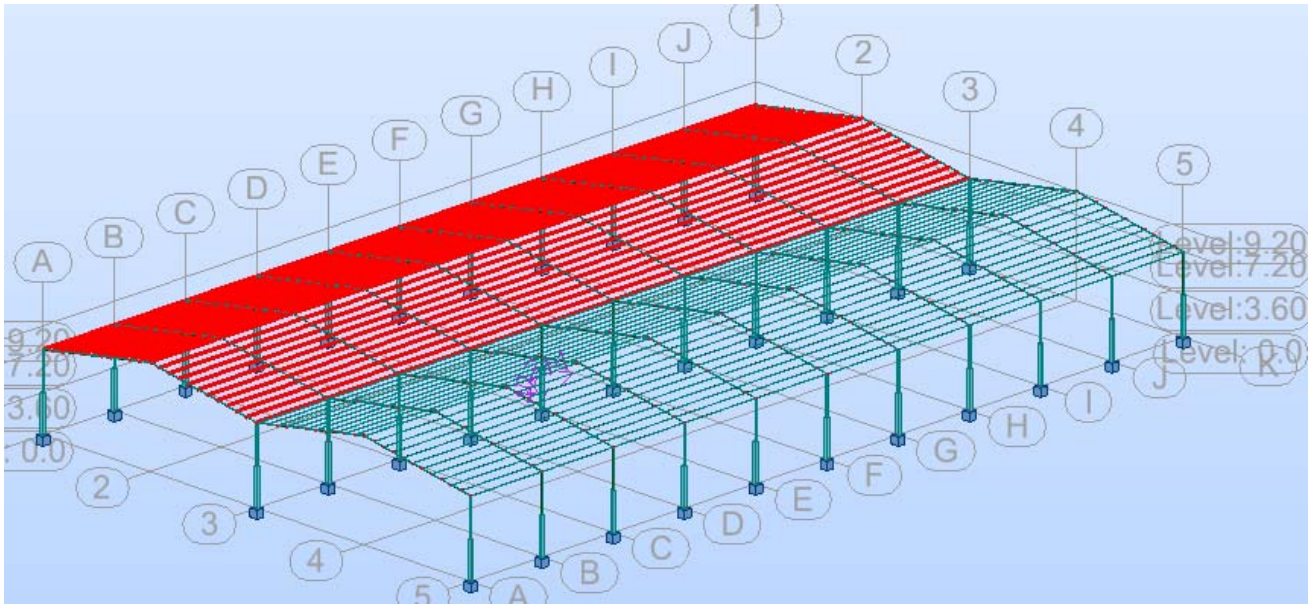


ចុច Edit menu > Select Special > Bars only……> CAE 40x3 > Icon  រួចចុច Close

- ចុច Edit menu > Edit > Vertical Mirror ….. រួចចុចត្រង់ចំនុច

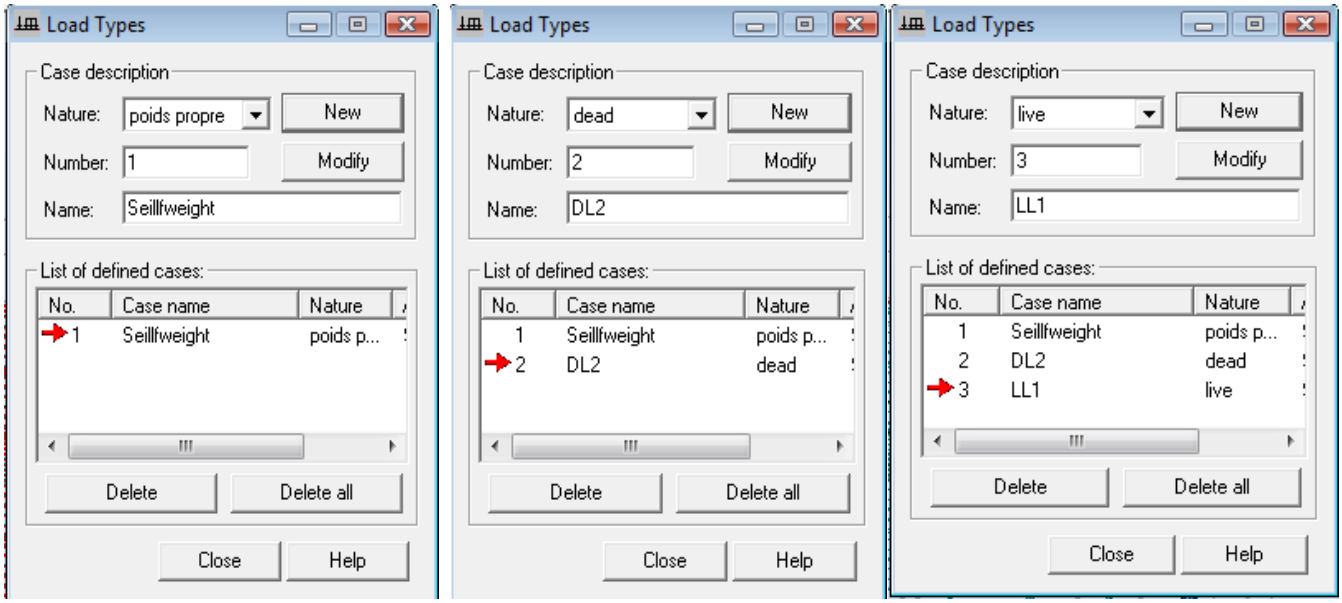






<u>១២. ការកំណត់បន្ថែមអំពីលើធ្នឹម:</u>

- Selfweight
- Dead Load (DL1)
- Live Load (LL1)
- CLose

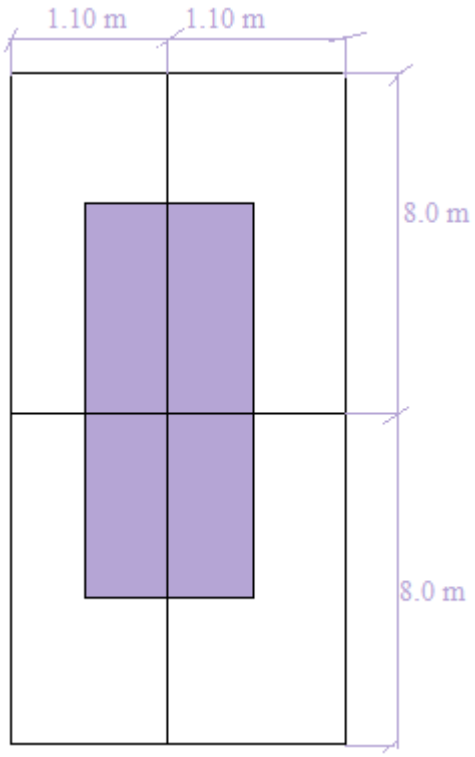


www.yerngshare.blogspot.com

១៣. ការកំណត់តំលៃបន្ទុក:

- Insulation + Steel sheet : 0.30 KN/m²
- Rafter : 0. 50 KN/m²
- Live load : 0.35 KN/m²

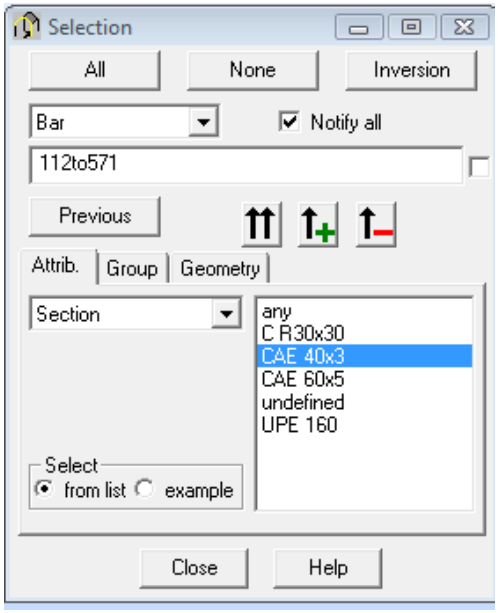
DL1 = (1.1 x 8.0) m² x (0.30 + 0.5) KN/m² = 7.04 KN
LL1 = (1.1 x 8.0) m² 0.35 KN/m² = 3.08 KN

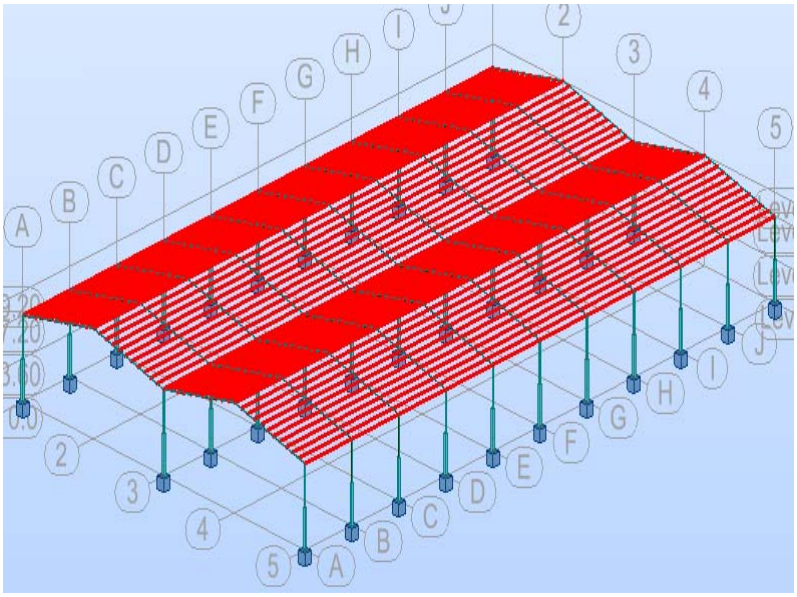


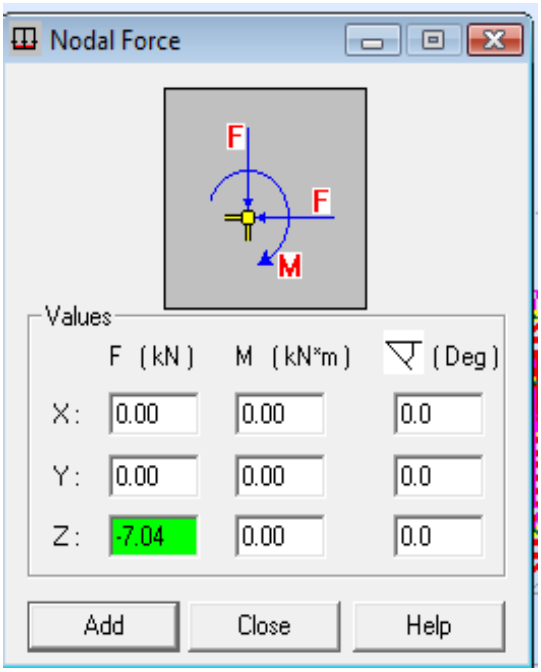
១៣. ការដាក់បន្ទុក (Load Definition)

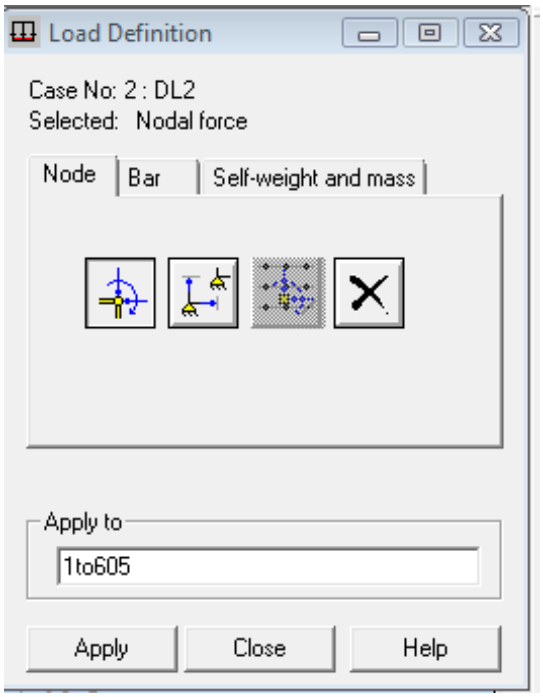
១៣.១ ការដាក់បន្ទុកថេរ (Dead load)

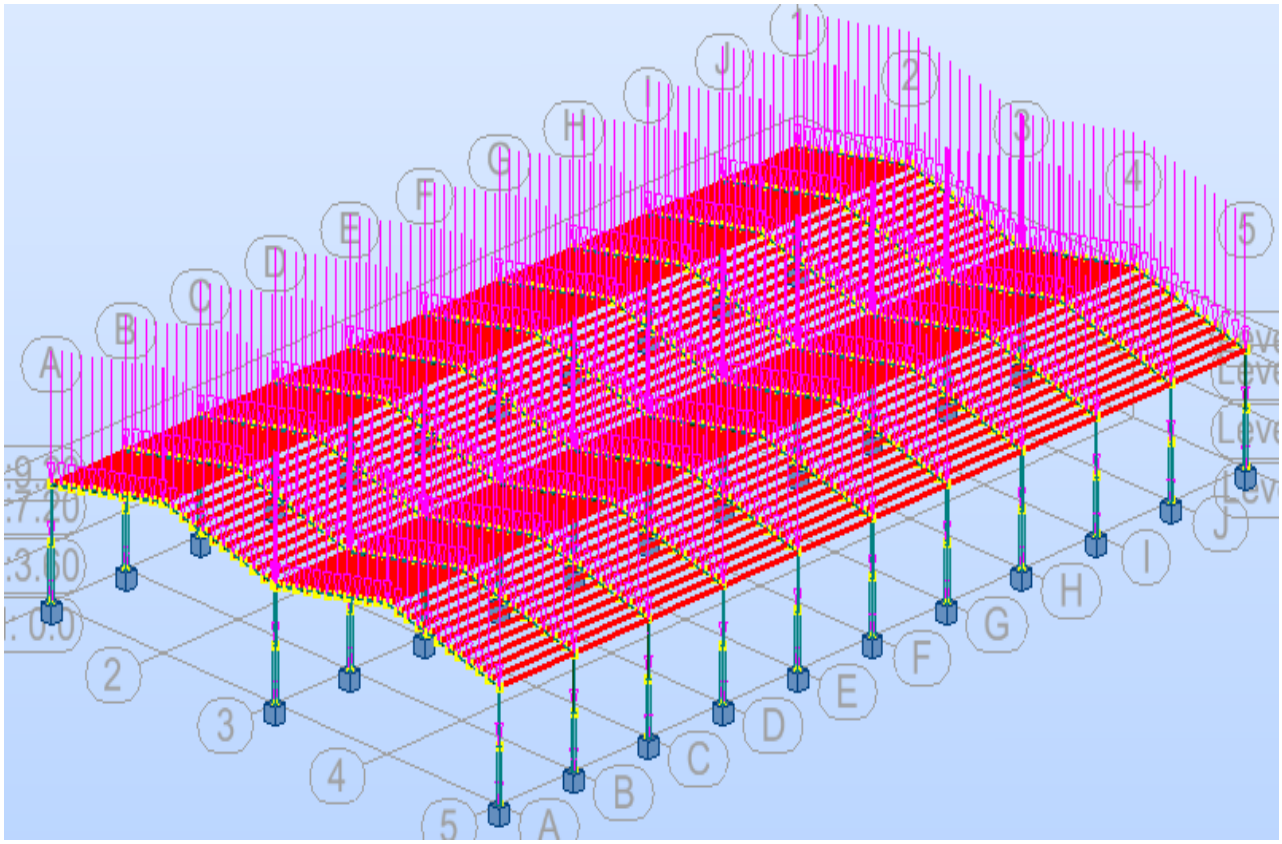
- Click Icon Load Definition
- Click Node
- បំពេញតំលៃក្នុងប្រអប់ Z = -7.04
- តែ Select Bars (CAE40x3) > Close
- ចុច Add
- Apply to វាយ All







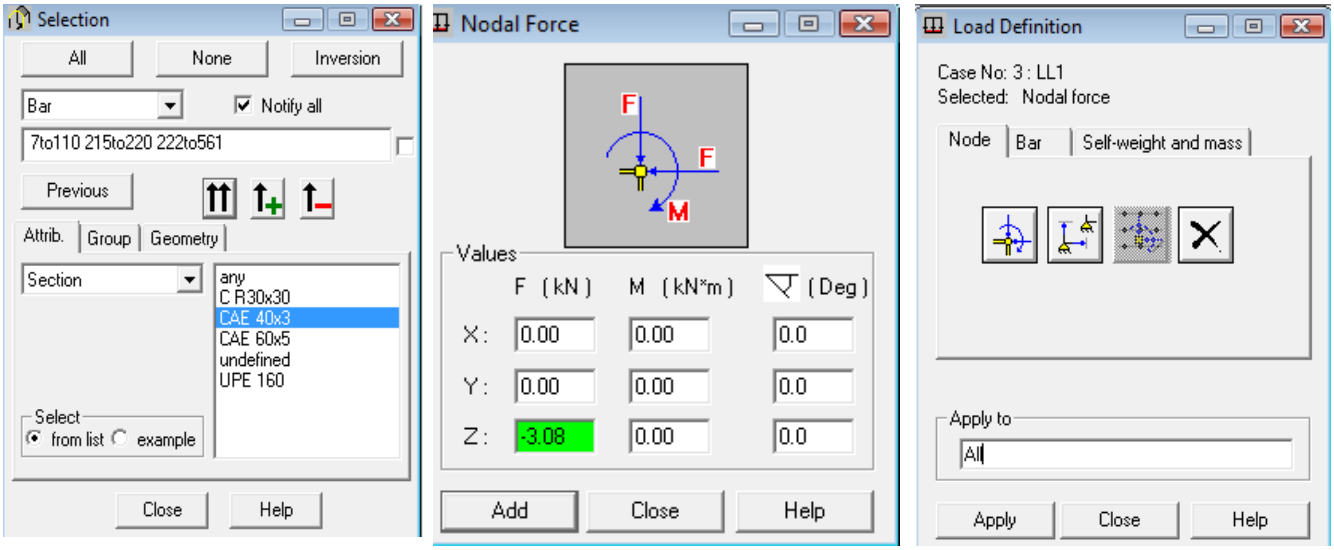




www.yerngshare.blogspot.com

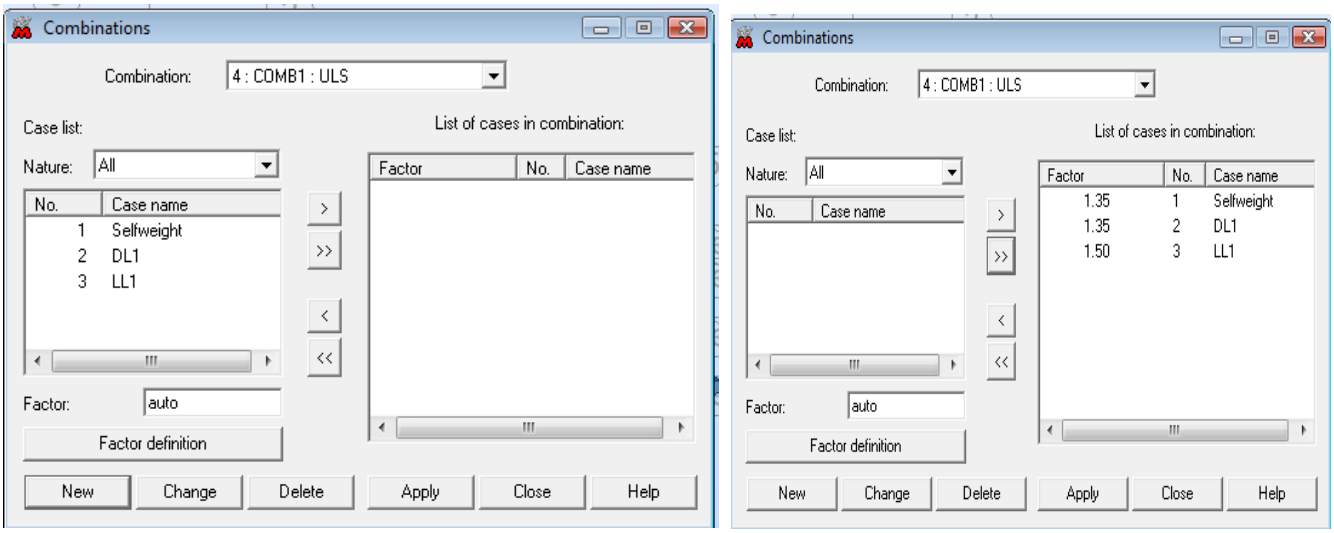
១៣.២. ការដាក់បន្ទុកអថេរ (Live load) ត្រូវធ្វើដូចដាក់បន្ទុកថេរខាងលើដែរ

- Click Icon Load Definition
- Click Node
- បំពេញតំលៃក្នុងប្រអប់ Z = -3.08
- តែ Select Bars (CAE40x3) > Close
- ចុច Add
- Apply to វាយ All



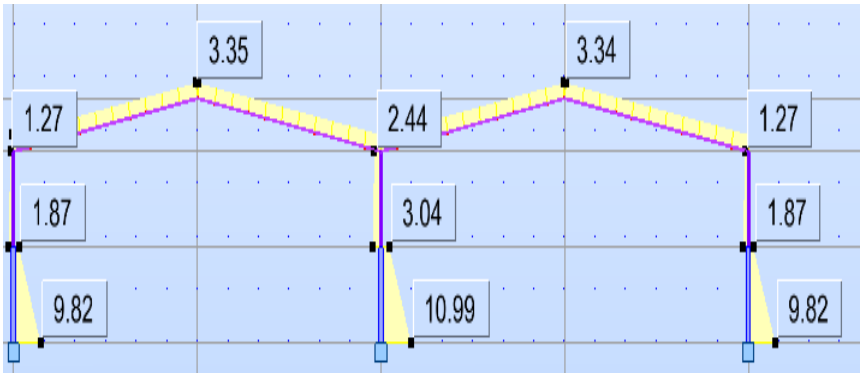
១៣.៣ ការប្រមូលបន្ទុក (Load Comination)

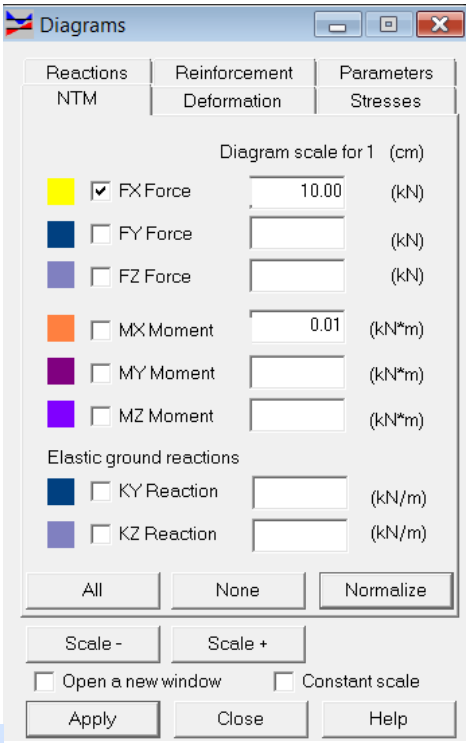
- ចូរ Loads menu > Manual Combinations > Next > Apply > Close



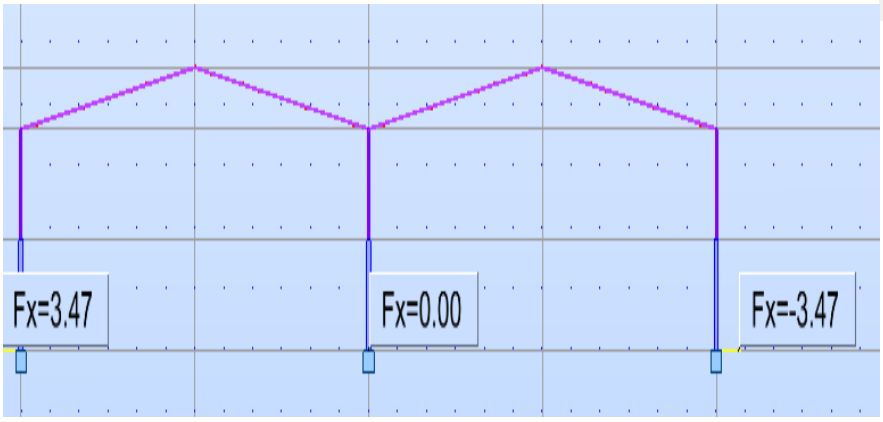
+ បន្ទាប់មក ត្រូវចុច Calculations (Run)

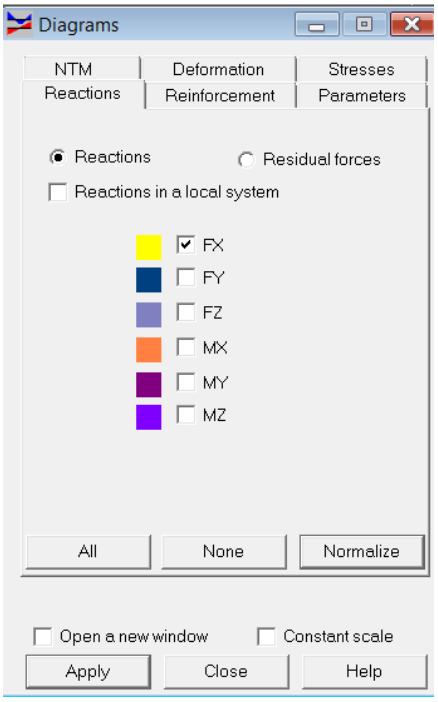
+ ចូរ Results > Diagrams for bars…..> NTM > Tick FX Force > Click Command button Apply



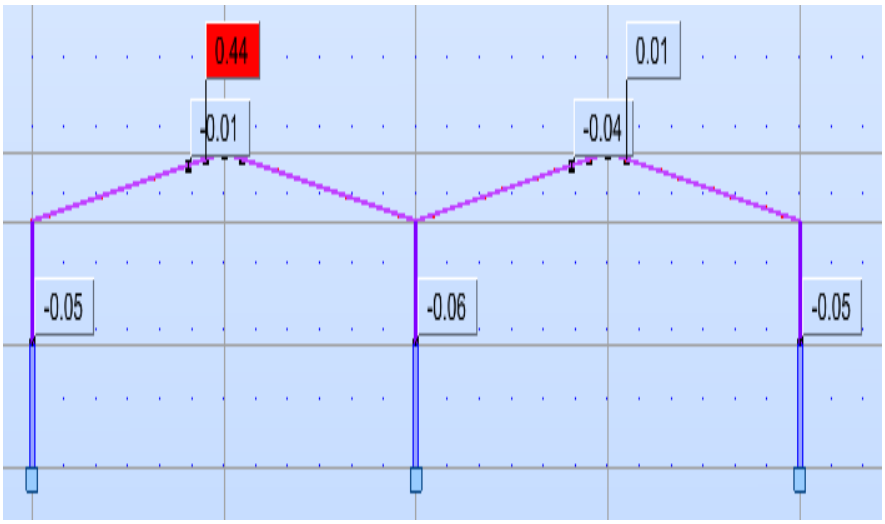


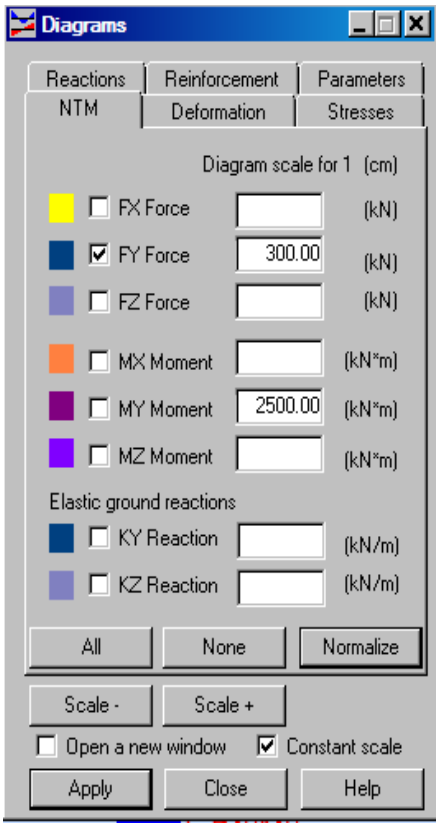
លទ្ធផល Diagrams **បង្ហាញពីកំលាំងតាមអ័ក្ស** X (FX Force)



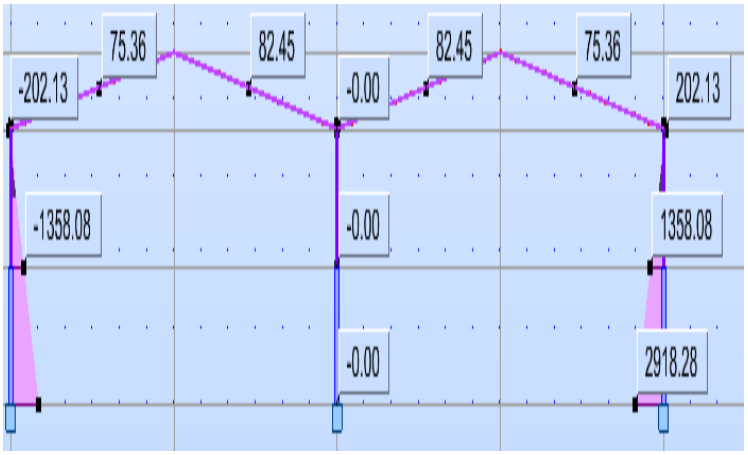


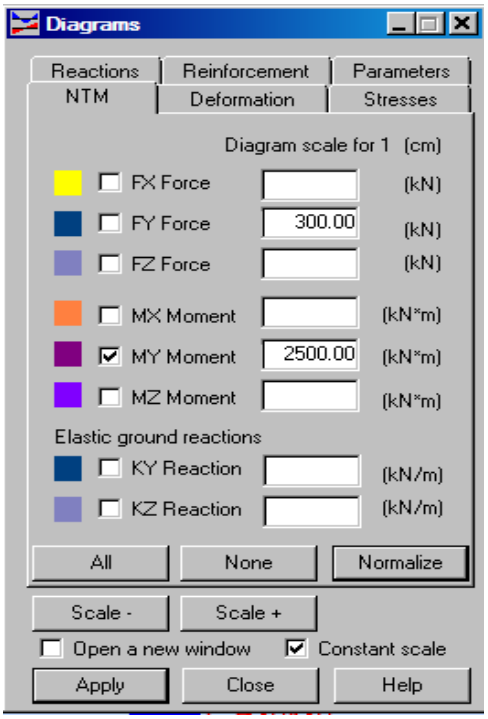
លទ្ធផល Diagrams **បង្ហាញពីប្រតិកម្មទំរតាមអ័ក្ស** X (FX)





លទ្ធផល Diagrams **បង្ហាញពីកំលាំងតាមអ័ក្ស** Y (FX Force)



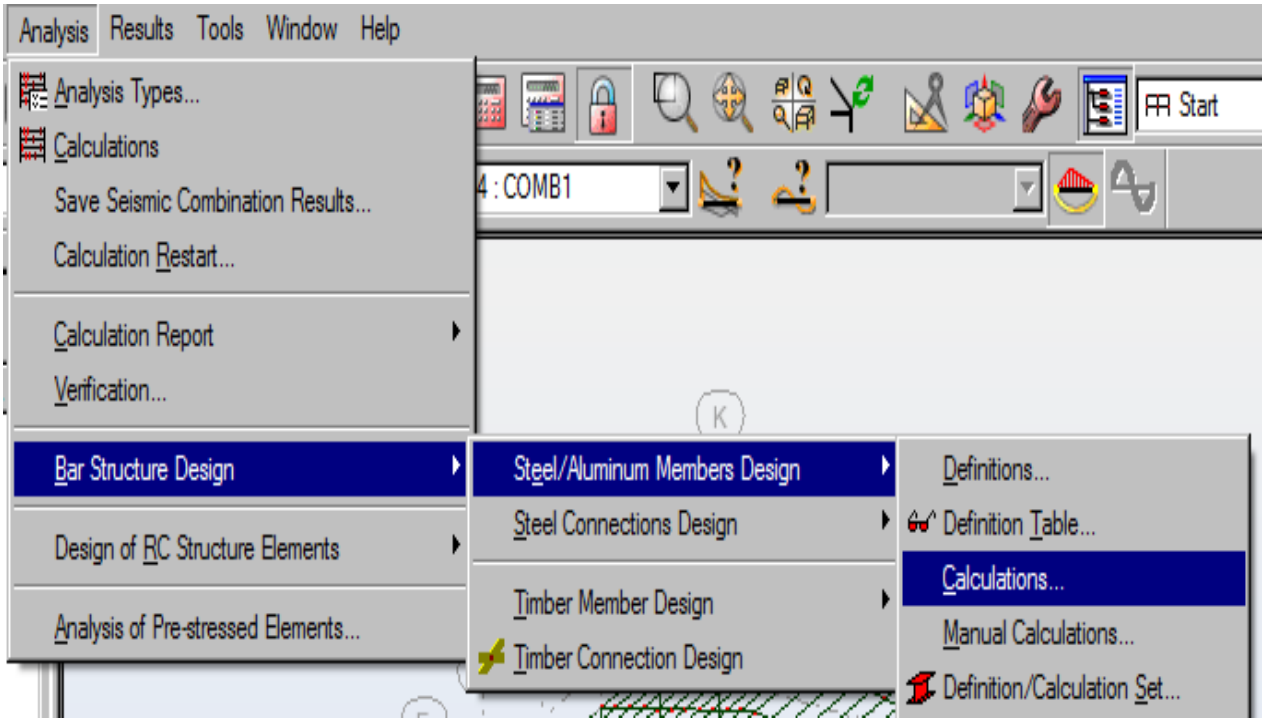


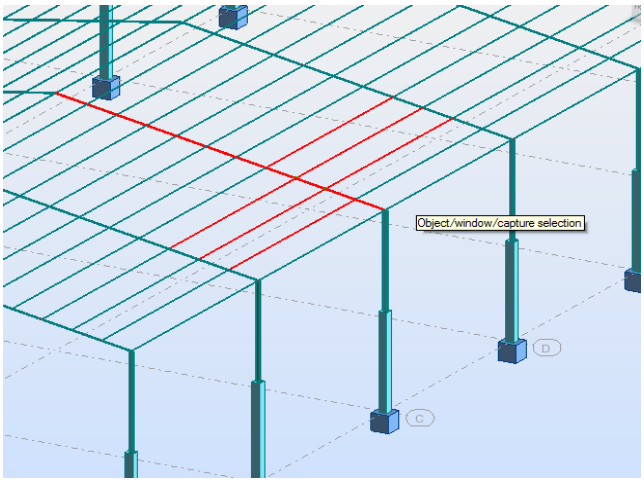
លទ្ធផល Diagrams **បង្ហាញពីម៉ូម៉ង់តាមអ័ក្ស**Y (My Moment)

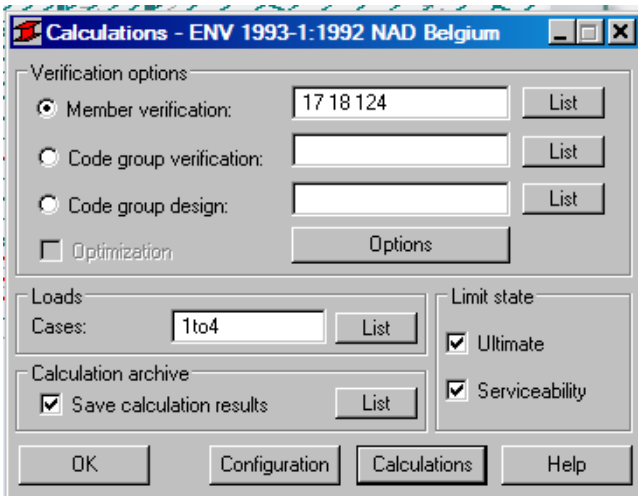
www.yerngshare.blogspot.com

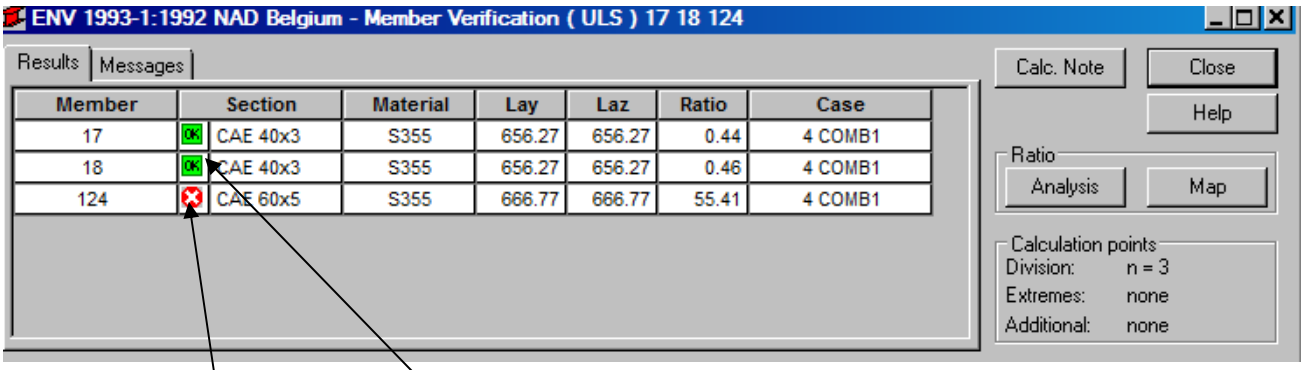
វិភាគមុខការត់ជែក (Analysis)

- Select Joint > Analysis > Bar structure design > Steel/ Aluminum members design > Calculation



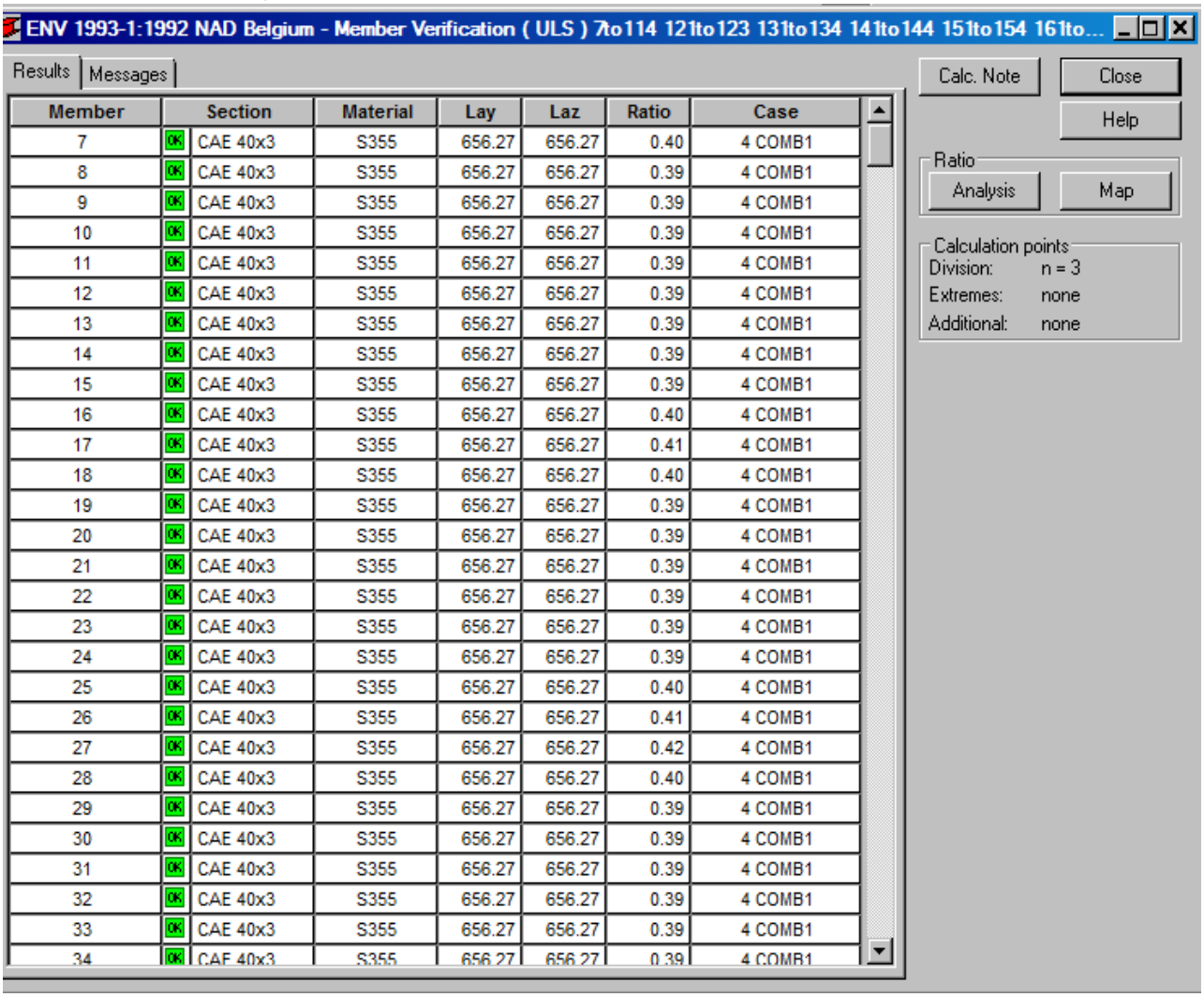






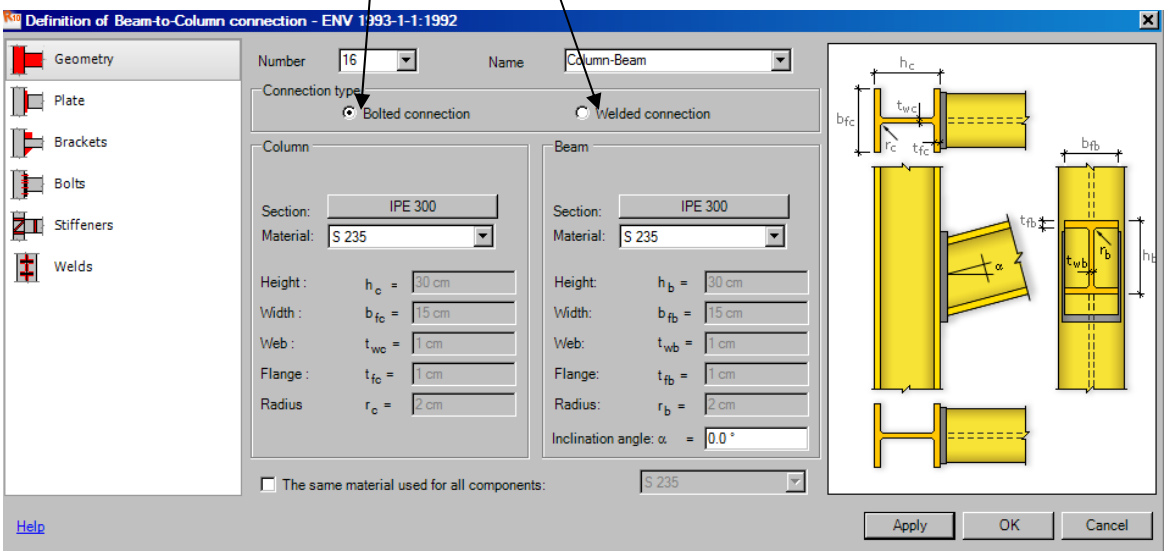
លទ្ធផល

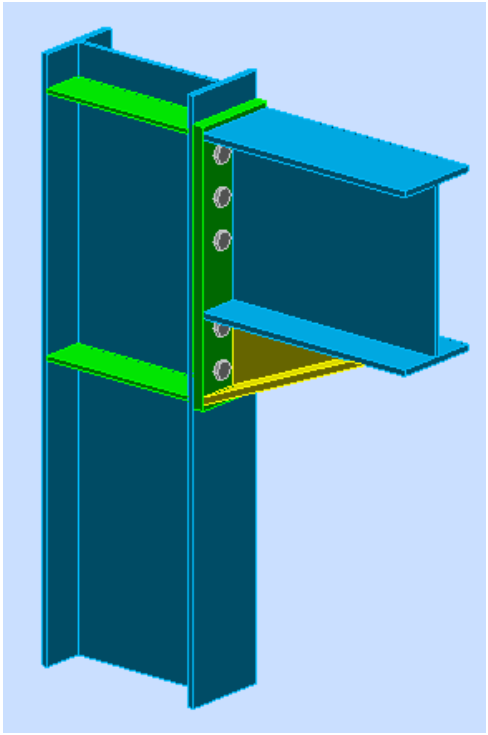
- សញ្ញាពណ៌ខ្សែរ មានន័យថាជាប់
- សញ្ញាពណ៌ខ្សែរ មានន័យថាមិនជាប់

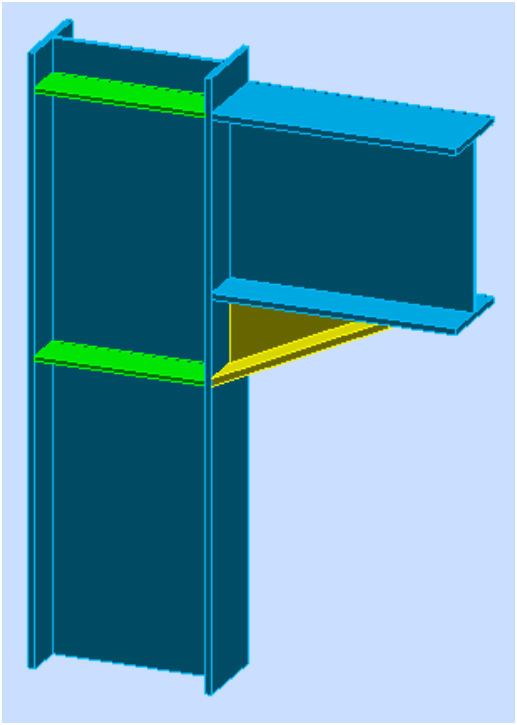


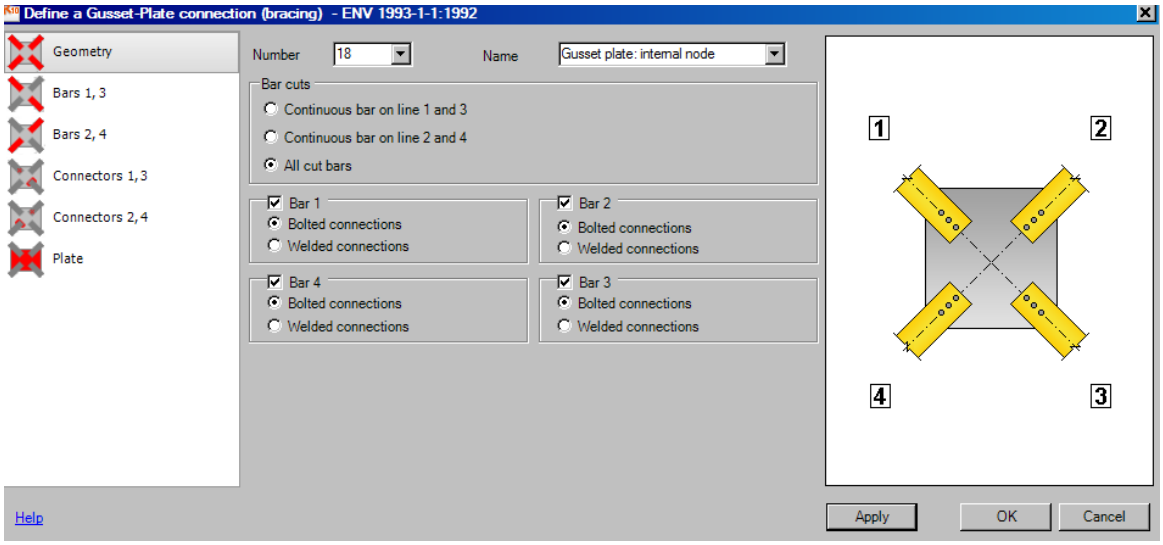
តំណរមានពីរប្រភេទ

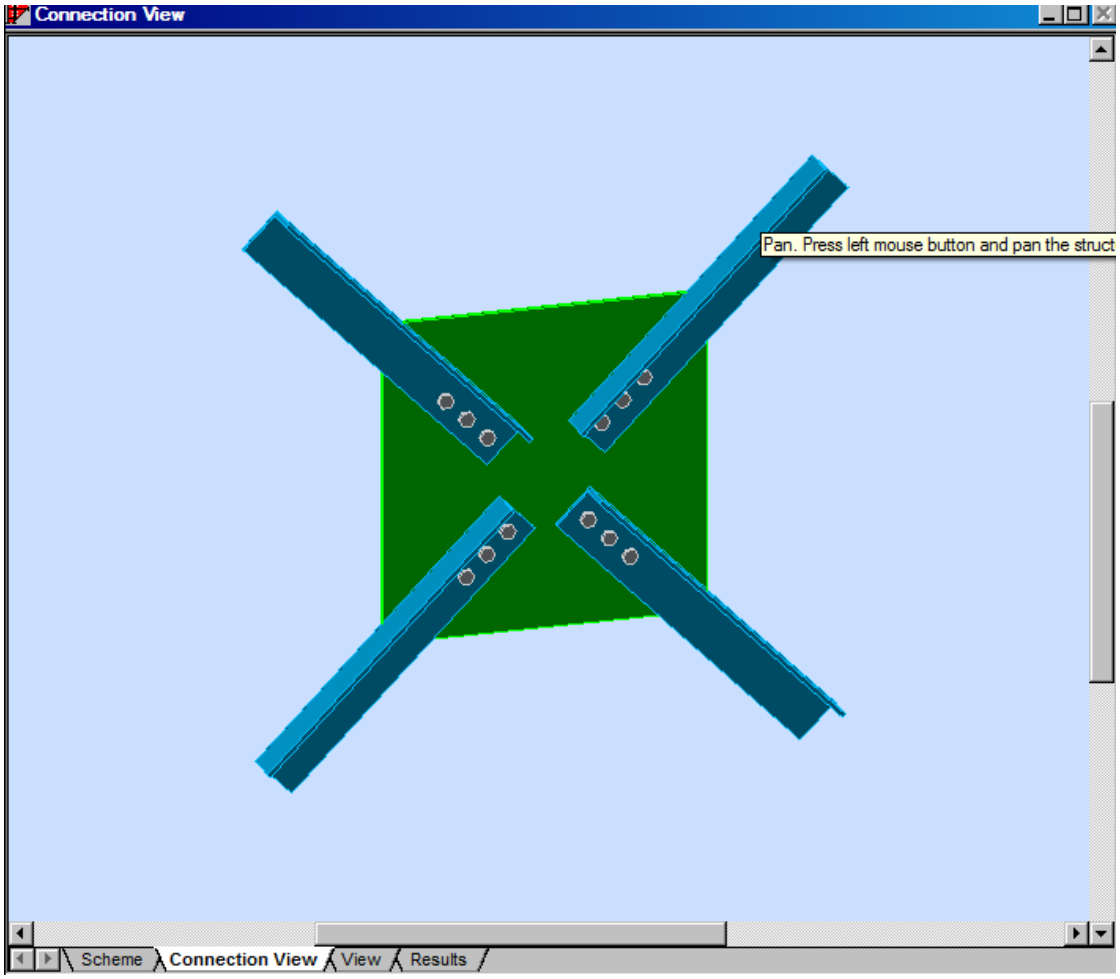
- តំណរប្រើប៊ូឡុង (Bolted Connection)
- តំណរផ្សារ (Welded connection)











www.yerngshare.blogspot.com